Commands

Run CSV Editor
Run Calculator
Run Clock
Run Control Panel
Run File Manager
Run Notepad
Run Picture Editor
Run Schematic Capture
Run Text Editor
Run User Program 1
Run User Program 2
Run User Program 3
Run User Program 4
Run Windows Setup
Save Environment Configuration File
Screen Redraw
Select All
Select Inside Area
Select Outside Area
Send To Back
Send To Back Of
Setup Autopan
Setup Printer
Setup User Programs
Shift Cursor Down
Shift Cursor Left
Shift Cursor Right
Shift Cursor Up
Show De-Morgan
Show IEEE
Show Normal
Toggle Command Status Bar
Toggle Drawing Toolbar
Toggle Hidden Pins
Toggle IEEE Toolbar
Toggle Main Panel
Toggle Main Toolbar
Toggle Scroll Bars
Toggle Selection
Toggle Single Object
Toggle Snap Grid
Toggle Status line
Toggle Visible Grid
Undo
Update Schematic Editor
Using Help
Window Arrange Icons
Window Cascade
Window Close All
Window Tile
Workspace Options
Zoom 100%
Zoom 200%
Zoom 400%
Zoom 50%
Zoom All

Zoom In
Zoom Out
Zoom Pan
Zoom Point
Zoom Window
Zoom Workspace

File Open

Summary Open and load a component library file.

To Launch F O - 

Purpose The file open command is used to load and display the contents of a component library file into a new document window, ready to visually browse or edit.

Outcome Once the library file has been selected it will load into a new document window and all the workspace preferences will be restored. This is because Schematic Library Editor stores the workspace options in the library file whenever you save the file.

Comments Schematic Library Editor can load any number of library files into separate windows using the Multiple Document Interface (MDI). Schematic Library editor will load Protel Schematic (Binary and ASCII) library files as well as OrCAD STD III and STD IV.

When no document file is open, the Menu bar displays three options: File, Info and Help.

You can abort the drawing (or redraw) of the document window, at any time, by pressing the spacebar. This allows you to move directly to another menu command or Tool button without waiting for the entire screen redraw to be completed.

Procedure To open a previously created file:

1. Choose Open from the File menu.

2. Type the file name (include the full path, if different from the path listed after Directory)

3. Click OK to open the file.

You can also double-click on the desired file name in the Files window, if any. To change directories, click under any of the options listed in the Directories window.

File Merge

Summary Merge components from another library to current library file.

To Launch Alt F M

Purpose The File Merge command is used to copy the contents of another library file to the library in the current workspace. This command is used to create customized libraries.

Outcome Once the source library file has been selected, all of the components in this file will be copied to the current library in the workspace.

Comments Merging component libraries simply copies all of the contents of one library file to another file. Components are not deleted or renamed from any of the files. Duplicate components may be removed by using the Remove Duplicates command from the Component menu.

Procedure To merge components from another library file into the current library.

1. Choose Merge from the File menu.

2. Type the source file name (include the full path, if different from the path listed after Directory)

3. Click OK to merge the contents of the chosen file to the current library.

You can also double-click on the desired file name in the Files window, if any. To change directories, click under any of the options listed in the Directories window.

See also

Copy Component
Remove Duplicates

File New

Summary Create a new and empty component library file.

To Launch F N

Purpose The File New command is used to create a new blank component library.

Outcome When you choose the File New command, an empty document window is displayed and becomes the current window with the title "LIB_#."

Comments You can change and define various workspace settings, such as workspace orientation, background color, border color and other settings by choosing Workspace from the Options menu.

Procedure To create a new component library:

1. Choose New from the File menu.

See also

Workspace Options

Copy Component

Summary Copy component from current library to another library file.

To Launch Alt C C

Purpose The Copy Component command is used to copy a component from the current library file to another library file. This command is used to create customized libraries.

Procedure To copy the current component into another library.

1. Choose Copy Component from the Component menu.

2. Choose the destination library file.

3. Click OK to copy the component in the current library document window to the chosen destination library file.

You can also double-click on the desired destination library file name.

See also

File Merge
Move Component

Move Component

Summary Move component from current library to another library file.

To Launch Alt C M

Purpose The Move Component command is used to move a component from the current library file to another library file. This is similar to the Copy Component command, except that the component is physically removed from the current library and placed in the destination library. This command is used to create customized libraries.

Procedure To Move the current component into the another library.

1. Choose Move Component from the Component menu.

2. Choose the destination library file.

3. Click OK to move the component in the current library document window to the chosen destination library file.

You can also double-click on the desired destination library file name.

See also

Copy Component

File Save All

Summary Save all loaded component library files.

To Launch Alt F L

Purpose The File Save All command can be used to save all of the component library files in all currently opened document windows.

Outcome When you select the File Save All command, this will save the contents of all the library document windows to the Schematic Library Editor binary file format, using the existing paths and file names.

Procedure To save all library files which are currently opened:

1. Choose Save All from the File menu.

See also

File Save
File Save As

Run Schematic Capture

Summary Launch and Switch to the Schematic Sheet Editor.

To Launch T H - 

Purpose The Run Schematic Capture command is used to launch and switch to the Advanced Schematic Editor application window.

Comments The Run Schematic command can also be activated by clicking Place in the Main Panel window.

Procedure To switch to the Schematic Editor:

1. Choose Run Schematic from the File menu.

   If the Schematic Editor has been previously loaded then the focus will switch to the Schematic Editor application window, otherwise the Schematic Editor will be launched, loading previously opened schematic worksheet files.

See also

New Part
Update Schematic Editor

Update Schematic Editor

Summary Update components in all opened schematic worksheet files.

To Launch Alt F U

Purpose The Update Schematic Editor command is used to automatically update/replace matching parts in opened schematic worksheet files with the component in the current document window.

Procedure To automatically update modified schematic components:

1. Choose Update Schematic from the File menu.

See also

File Exit

Summary Quit from Schematic Library Editor.

To Launch F X

Purpose The File Exit command is used to close down all opened component library files, and end the current work session of the Library Editor.

Outcome When the File Exit command is selected, this quits the current session of Schematic Library Editor and prompts to save modified files.

Procedure To quit the current session of Schematic Library Editor:

1. Choose Exit from the File menu.

2. If any document window has been modified since last saving, a dialog box will be displayed that prompts to save the library file(s). Confirm to save the file(s) or select Cancel to ignore the File Exit command.

Hot Keys

Summary Setup Hot Key assignments.

To Launch O H

Purpose The Options Hot Keys command is used to assign keyboard keys and mouse button clicks to the Library Editor command processors, such as "Place Pins," "Show De-Morgan," "Toggle Snap Grid" or any other command.

Comments There are three other buttons in this dialog box; Load is used to open a previously saved hot key assignments file. Save is used to save the current hot key assignments to a file. Finally, Defaults is used to restore the current hot key assignments to the Library Editor default settings which are listed at the back of this reference.

Procedure To assign keyboard keys and mouse button clicks to command processes;

1. Choose Hot Keys from the Options menu.

2. Select a command from the Menu Commands window.

3. Select a keyboard key or mouse click such as "Right Click" from the Keys window. You can also use a combination of CTRL and SHIFT.

4. Click the Assign button to accept the hot key assignment.

5. Assign more commands to hot keys or click OK to close the Hot Keys dialog box.

See also

Preferences

Summary Setup System Preferences.

To Launch O P

Purpose The Options Preferences command is used to define various system settings, such as auto backup time intervals, cursor shape, auto pan, delete confirmation, and other options that apply to all document windows.

Outcome The following are options that can be set within the preferences dialog box.

Selections - The default color for selected objects can be changed by clicking in the color box and choosing another color from the Color Selector dialog box. Schematic Library Editor will display all the available colors that your graphics card and monitor supports.

Grid Color - The visible can be assigned a default color. To assign new color to the visible grid, click in the color box to open the Color Selector dialog box. Schematic Library Editor will display all the available colors that your graphics card and monitor supports.

Cursor Type - Three options are available for the physical (or workspace) cursor, a large 90 degree cross, small 90 degree cross or a 45 degree cross. The workspace cursor is displayed when making selections, moving or placing objects. The workspace cursor snaps to each grid point if the snap grid is turned on, making it easier to control objects being placed or moved. The system 'arrow' is able to select objects independently of the snap grid being on or off, which makes selection easier when zoomed-out or when dealing with off-grid objects. The default cursor type is the small 90 degree cross.

Visible Grid - The visible grid can be displayed as lines or dots.

Undo Limit - This option allows the user to set the number of Undo/Redo levels. The default is 50. This allows the user to backtrack through (and restore) 50 individual operations. The legal range is 0-16000. Because each operation must be stored in memory to enable Undo/Redo, setting a higher number may result in performance deterioration, while editing. If you notice Advanced Schematic slowing down after using the application for a while, try selecting a lower number. The more memory you have installed in your machine, the higher the limit you can set without hurting the performance of your system. Setting this option to 0 will clear the Undo/Redo memory stack and disable the Undo/Redo command.

Dialog Text Font - The default font that is used for all dialog boxes can be changed with this command. The Library Editor must be closed and re-opended in order for this new font to change.

Snap To Center - If the Snap to Center option is enabled the cursor will jump to the center of an object when it is moved.

Save Defaults - When this option is enabled, user settings, preferences and defaults are saved in an initialization file when exiting Schematic Library Editor. Save Defaults is enabled by default.

Auto Backup Time - Sets the interval (in minutes) for generating an automatic backup

of the active library file. To disable this feature, type 0. The default interval is 30 minutes.

Clipboard Reference - When this option is enabled, you will be prompted to select a reference point when copying and cutting selected objects to the clipboard.

Use Printer Fonts on Screen - When this option is enabled the fonts that have been used on the screen will be substituted for the closest matching fonts that the current selected printer supports. For example, if a vector plotter is selected and true type fonts are used on the schematic sheet, all the true type fonts on the screen will be displayed with the closest matching font that the plotter supports. This option is disabled by default.

Auto Zoom - When enabled, redraws the workspace with the object centered in the window (not active when using Find command or Jump from the Browser). For example, when using the Window-Tile command this option re-draws the whole sheet centered in the window.

Convert Special Strings - When this option is enabled, the special strings that have been placed onto the worksheet, for example, ".DATE", are converted to show their true representations, in this case the current system date would be displayed. The following special strings may be placed on the schematic worksheet, .DATE, .TIME, .DOC_FILE_NAME, .organization, .address1, .address2, .address3, .address4, .sheetnumber, .sheettotal, .title, documentnumber and .revision.

Procedure To setup the system preferences:

1. Choose Preferences from the Options menu.

2. Type in new system settings.

3. Click OK to accept the new settings.

See also

Preferences

Toggle Main Panel

Summary Turn Main Panel on or off.

To Launch O A - 

Purpose The Toggle Main Panel command displays or hides the main panel, depending upon its current state.

Comments If the Main Panel is currently displayed, a check mark will be visible to the left of Main Panel on the Options menu.

Procedure To turn the Main Panel on or off:

1. Choose Main Panel from the Options menu.

Toggle Main Toolbar

Summary Turn the Main Toolbar on or off.

To Launch O M

Purpose The Toggle Main Toolbar command is used to turn the Main Toolbar on or off.

Comments A listing of all the Main Toolbar buttons and commands can be found in the Tool Bars section, at the back of this reference manual.

Procedure To turn the Main Toolbar on or off:

1. Choose Main Toolbar from the Options menu.

See also

Toggle IEEE Toolbar
Toggle Drawing Toolbar
Toggle Main Panel

Toggle IEEE Toolbar

Summary Turn the IEEE Toolbar on or off.

To Launch O I - 

Purpose The Toggle IEEE Toolbar command is used to turn the IEEE Toolbar on or off.

Procedure To turn the IEEE toolbar on or off:

1. Choose IEEE Toolbar from the Options menu.

See also

Toggle Main Toolbar
Toggle Drawing Toolbar
Toggle Main Panel

Toggle Drawing Toolbar

Summary Turn the Drawing Toolbar on or off.

To Launch O R - 

Purpose The Toggle Drawing Toolbar command is used to turn the Drawing Toolbar on or off.

Comments A listing of all the Drawing Toolbar buttons and commands can be found in the Tool Bars section, at the back of this reference manual.

Procedure To turn the Drawing Toolbar on or off:

1. Choose Drawing Toolbar from the Options menu.

See also

Toggle Main Toolbar
Toggle IEEE Toolbar
Toggle Main Panel

Toggle Status line

Summary Turn the Status Line on or off.

To Launch O T

Purpose The Toggle Status line command is used to turn the Status line on or off at the bottom of the Schematic Library Editor window. Turning off the Status line shows more screen region.

Comments The status line displays information about the current activity or mode. When a command is selected, the left side of the status line briefly describes the chosen command. The left side of the status line also indicates the current cursor position.

Procedure To turn the Status line on or off:

1. Choose Status line from the Options menu.

Popup Edit Menu

Summary Popup the Edit menu.

Purpose The Popup Edit Menu command is used to directly access the Edit menu at the current cursor position.

Procedure To popup the Edit menu at the current cursor position.

1. By default, press E.

You can change the default hot key by using the Options Hot Keys command.

See also

Hot Keys

Popup Place Menu

Summary Popup the Place menu.

Purpose The Popup Place Menu command is used to directly access the Place menu at the current cursor position.

Procedure To popup the Place menu at the current cursor position.

1. By default, press P.

You can change the default hot key by using the Options Hot Keys command.

See also

Hot Keys

Popup Select Menu

Summary Popup the Edit Select menu.

Purpose The Popup Select Menu command is used to directly access the Edit Select menu at the current cursor position.

Procedure To popup the Edit Select menu at the current cursor position.

1. By default, press S.

You can change the default hot key by using the Options Hot Keys command.

See also

Hot Keys

Popup De-Select Menu

Summary Popup the Edit De-Select menu.

Purpose The Popup De-Select Menu command is used to directly access the Edit De-Select menu at the current cursor position.

Procedure To popup the Edit De-Select menu at the current cursor position.

1. By default, press x.

You can change the default hot key by using the Options Hot Keys command.

See also

Hot Keys

Popup Zoom Menu

Summary Popup the Zoom menu.

Purpose The Popup Zoom Menu command is used to directly access the Zoom menu at the current cursor position.

Procedure To popup the Zoom menu at the current cursor position.

1. By default, press z.

You can change the default hot key by using the Options Hot Keys command.

See also

Hot Keys

Popup Options Menu

Summary Popup the Options menu.

Purpose The Popup Options Menu command is used to directly access the Options menu at the current cursor position.

Procedure To popup the Options menu at the current cursor position.

1. By default, press O.

You can change the default hot key by using the Options Hot Keys command.

See also

Hot Keys

Popup Help Menu

Summary Popup the Help menu.

Purpose The Popup Help Menu command is used to directly access the Help menu at the current cursor position.

Procedure To popup the Help menu at the current cursor position.

1. By default, press H.

You can change the default hot key by using the Options Hot Keys command.

See also

Hot Keys

Popup File Menu

Summary Popup the File menu.

Purpose The Popup File Menu command is used to directly access the File menu at the current cursor position.

Procedure To popup the File menu at the current cursor position.

1. By default, press F.

You can change the default hot key by using the Options Hot Keys command.

See also

Hot Keys

Popup Info Menu

Summary Popup the Info menu.

Purpose The Popup Info Menu command is used to directly access the Info menu at the current cursor position.

Procedure To popup the Info menu at the current cursor position.

1. By default, press I.

You can change the default hot key by using the Options Hot Keys command.

See also

Hot Keys

Popup Window Menu

Summary Popup the Window menu.

Purpose The Popup Window Menu command is used to directly access the Window menu at the current cursor position.

Procedure To popup the Window menu at the current cursor position.

1. By default, press w.

You can change the default hot key by using the Options Hot Keys command.

See also

Hot Keys

Information System Status

Summary Display system information.

To Launch I S

Purpose The Information System Status command is used to display system information, such as available disk space, free memory, date and time settings.

Comments *Free memory* - Amount of memory (in kilobytes) your system currently has free. If your computer is running under 386 enhanced mode, this will also include the amount of virtual memory available. This is the sum of your physical memory left and your disk space available on the drive where your swap file is set-up.

Total Disk Space - Total storage space of your hard disk in kilobytes.

Free Disk Space - Amount of disk storage space (in kilobytes) available on your hard disk.

System Date - The current date of your system. Use the Control Panel to change the system date.

System Time - The current time of your system. Use the Control Panel to change the system time.

System Resources - Resources currently available on your system, displayed in percentage.

Procedure To display the current system settings:

1. Choose System Status from the Info menu.

A dialog box will open displaying the current system settings.

Help About

Summary Display the version number and copyright of Schematic Library Editor.

To Launch H A

Purpose The Help About command is used to display the application name, version number and copyright information of Schematic Library Editor.

Outcome When you choose the Help About command a dialog box will open displaying the current version number and copyright information for Schematic Library Editor.

Comments This command could be used to find the current version number of Schematic Library Editor before calling technical support.

Procedure To display the version number of Schematic Library Editor:

1. Choose About from the Help menu.

See also

Help Basic Concepts

Summary Basic information about Schematic Library Editor.

To Launch H B

Purpose The Help Basic Concepts menu option displays an index of basic operational topics of Schematic Library Editor.

Outcome Once the Help Basic Concepts command is selected the Schematic Library Editor on-line help system is loaded and displays an index of basic operational topics.

Procedure To display the index of basic operational topics:

1. Choose Basic Concepts from the Help menu.

2. Then choose the specific topic from the help index list.

See also

Help Index

Help Commands

Summary Help information organized by menu structure.

To Launch H C

Purpose The Help Commands menu option displays the Schematic Library Editor index of menu commands, the topics offer access to detailed information about each command.

Outcome Once the Help Command option is selected the Schematic Library Editor on-line help system is loaded and displays an index of the menu commands available in Schematic Library Editor.

Procedure To display the index of menu commands:

1. Choose Commands from the Help menu.

2. Then choose the specific topic from the help index list.

See also

Help Index

Using Help

Summary Information about the Windows Help System.

To Launch H H

Purpose The Using Help command displays the introductory topic about how to access and use the on-line help system.

Outcome Once the Using Help command is selected the Schematic Library Editor on-line help system is loaded and displays instructions on how to use help.

Procedure To display the information on how to use the help system:

1. Choose Using Help from the Help menu.

2. Then choose the specific topic from the help index list.

See also

Help Index

Help Index

Summary Help system topic index.

To Launch F1 - H I - 

Purpose The Help Index menu option displays the cross-referenced topics available under the help system.

Outcome Once the Help Index command is selected the Schematic Library Editor on-line help system is loaded and displays the index of contents.

Procedure To display the Help index:

1. Choose Index from the Help menu.
2. Then choose the specific topic from the help index list.

Help Printing

Summary Information about generating hard copy output.

To Launch H P

Purpose The Help Printing menu option displays an index of printing and plotting topics.

Outcome Once the Help Printing command is selected the Schematic Library Editor on-line help system is loaded and displays an index of printing output topics.

Procedure To display the index of printing topics:

1. Choose Printing from the Help menu.

2. Then choose the specific topic from the help index list.

See also

File Print
Setup Printer
Help Index

Help Reference

Summary Schematic Library Editor Reference.

To Launch H R

Purpose The Help Reference menu option displays an index of additional topics covering some of the automation features of Schematic Library Editor.

Outcome Once the Help Reference command is selected the Schematic Library Editor on-line help system is loaded and displays an index of reference topics.

Procedure To display the index of Reference topics:

1. Choose Reference from the Help menu.

2. Then choose the specific topic from the help index list.

See also

Help Index

Window Tile

Summary Tile all open schematic windows.

To Launch Shift F4 - Alt W T

Purpose The Window Tile command is used to arrange all open library document windows so that they are all visible and do not overlap.

Outcome Once the Tile command has been chosen all the open and non-minimized library document windows will be re-sized and arranged so that they are all visible and do not overlap.

Procedure To Tile all the non-minimized library document windows:

1. Choose Tile from the Window menu.

See also

Window Cascade

Window Cascade

Summary Cascade all open schematic windows.

To Launch Shift F5 - Alt W C

Purpose The Window Cascade command is used to arrange all open library document windows so that they all overlap and each document title bar is visible.

Outcome Once the Cascade command has been chosen all the open and non-minimized library document windows will be re-sized and arranged so that they all overlap and each document title bar is visible.

Procedure To Cascade all the non-minimized library document windows:

1. Choose Cascade from the Window menu.

See also

Window Tile

Window Arrange Icons

Summary Arrange all minimized open library document windows.

To Launch Alt W I

Purpose The Window Arrange Icons command is used to arrange the icons of all minimized library documents across the bottom of the Library Editor main window.

Procedure To Arrange all the minimized library document icons:

1. Choose Arrange Icons from the Window menu.

See also

Window Cascade

Window Close All

Summary Close all open schematic windows.

To Launch Alt W L

Purpose The Window Close All command is used to close all library document windows and icons.

Procedure To close all the opened library document windows and icons:

1. Choose Close All from the Window menu.

2. If any library has been modified since last saving, a dialog box will be displayed that prompts to save the file(s). Confirm to save the file(s) or press Cancel to ignore the Close All command.

See also

File Close

File Print

Summary Print the current document with the last set printer options.

To Launch Alt F P

Purpose The File Print command is used to print or plot the active document window, using the current printer/plotter settings defined in the Printer Setup dialog box.

Outcome The library will be sent to the printer or plotter. A small dialog box is displayed showing the output device and page count. The status line shows the percentage complete.

Comments If you have selected a file as the output port, then you will be asked for an output file name.

Procedure To print the current document window:

1. Choose Print from the File menu.

See also

Setup Printer

Setup Printer

Summary Printer and page setup.

To Launch Alt F I - 

Purpose The Setup Printer command is used to change printers and to control printing options on the selected printer. This command is also used to setup the scale, color mode and margins of your workspace.

Outcome The following are options that can be set within the Printer Setup dialog box.

Select Printer - The available output device options will include those that have been installed using the Windows Control Panel, (see your *Microsoft Windows User's Guide* for details).

Batch Type - Allows you to choose between Single Component or All Components in the library.

Color Mode - Allows you to choose between Color or Monochrome modes. Color mode allows you to print the workspace in color, if you are using a color-capable output device supported by a Windows 3 driver. Monochrome will print the workspace in a single color, regardless of the color capabilities of the output device.

Margins - Allows you to define the Top, Bottom, Left and Right margins.

Scale - Allows you to enlarge or reduce the size of your workspace by specifying a scale factor. If you wish to enlarge the workspace by 200% set the Scale to 2.000. The default value is 1.000.

Scale to Fit Page - The workspace will be expanded/contracted to fit within the defined margins for the current printer page size.

Comments Schematic Library Editor printing and pen-plotting are handled similarly to other Windows 3 applications. Windows manages the printing (or plotting) process and provides a range of raster and Postscript printer drivers and vector plotter drivers. These range from 9 pin dot matrix printers and multi-pen plotters, to high-resolution, color Linotronic imagesetters.

Procedure To setup your printer or plotter configuration:

1. Choose Setup Printer from the File menu.

2. Setup the output options in the Printer Setup dialog box.

3. Press the OK button to accept the printer settings and return to the editor without printing the document or;

4. Press the Print button to execute the File Print command.

The Preview button is used to visually check the orientation, positioning, margins and number of pages, before sending the file to the printer.

See also

File Print

File Close

Summary Close the current window.

To Launch Alt F C

Purpose The File Close command is used to close the current library document window.

Procedure To close the current library document window:

1. Choose Close from the File menu.

2. If the library document window has been modified since last saving, a dialog box will be displayed that prompts to save the library file. Confirm to save the file or select Cancel to ignore the Close command.

See also

Window Close All

File Save

Summary Save current library with same file name.

To Launch Alt F S - 

Purpose The File Save command is used to save the current library document window.

Outcome When you select the File Save command, this will save the contents of the current library document window to the Schematic Library Editor binary file format using the existing paths and file names.

Comments If the schematic library has been created with the File New command and has not been previously saved, the Save File As dialog box will open. You can save the file to a new path, new name or format. A backup file of the original file with an extension ".bak" is made before the new file is saved.

Procedure To save the current library:

1. Choose Save from the File menu.

See also

File Save All
File Save As

File Save As

Summary Save current component library with new file name.

To Launch Alt F A

Purpose The File Save As command can be used to save the active library file to a new path, new name or format.

Outcome When you select the File Save As command, this will open the Save File As dialog box, once you click OK the file will be saved to the new file name and/or directory that you specified.

Comments There are two file format options:

Protel Binary This is the default setting - an efficient format that enables Schematic Library Editor to open or save files more quickly than the OrCAD or text version.

Protel Text Protel's published (ASCII) text format for library files. The format of this file is described in the on-line help. ASCII files are less efficient than the binary version, but allow the user direct access to the design database for translation into other formats or other manipulation. If you open a Protel Text format file in Schematic Library Editor, it will be automatically saved in the default binary format when you choose File Save.

Procedure To save the current library document window to a new path, file name or format.

1. Choose Save As from the File menu.

2. Type a new file name for the component library. To change the output format of the file, choose one of the options listed in the File Types list.

3. Click OK to save the file.

To change directories, click under any of the options listed in the Directories window.

See also

File Save
File Save All

Undo

Summary Undo previous command.

To Launch Alt BackSpace - Alt E U - 

Purpose The Undo command is used to restore a library component to its state previous to the last operation. Multiple levels of undo are supported.

Outcome Reverses the effect of the previous edit(s), step-by-step, back to the number of operations specified under the Undo Limit option (Options-Preferences command).

Comments Schematic Library Editor stores operations as a stack in memory. Depending on the amount of memory available, you will eventually experience a noticeable decline in performance as the Undo/redo stack fills with large scale changes, such as global edits.

Procedure To undo the last operation:

1. Choose Undo from the Edit menu.

See also

Redo

Redo

Summary Redo previous undo command.

To Launch Ctrl BackSpace - Alt E R - 

Purpose The Redo command is used to restore changes made by the last Undo command. Multiple levels of Redo are supported.

Procedure To redo the last operation:

1. Choose Redo from the Edit menu.

See also

Undo

Cut

Summary Copy selected objects to clipboard and remove from workspace.

To Launch Shift Delete - Alt E T - 

Purpose The Edit Cut command is used to clear the objects that are currently selected in the workspace and copies it to the (internal) Schematic Library Editor clipboard (not the standard Windows clipboard). The Edit Paste command can be used to place the selection back into any open component library document window.

Outcome Removes the selected item(s) from the workspace and keeps a copy of the selection in the clipboard. The clipboard holds the last selection only, each time you use the Cut or Copy command, you overwrite the previous selection, stored in the clipboard.

Comments The Clipboard can be used to paste the selection to other Windows applications that support .WMF (Windows MetaFile) format.

Procedure To cut the current selection from the active window:

1. Make sure that the current selection includes only those objects you wish to cut;

You can use Edit De-Select to de-select objects that are not to be removed and then use Edit Select to select the objects that will be removed. You can also use the shortcut SHIFT+LEFT MOUSE to add objects to the current selection or to de-select any selected objects.

2. Choose Cut from the Edit menu. (shortcut: SHIFT+DELETE);

You will be prompted to select a reference point if the Clipboard Reference option is enabled in the Preferences dialog box. A reference point is a coordinate relative to the selected object(s). If the clipboard reference option is off then the current cursor position is used as the reference point.

See also

Copy
Paste

Copy

Summary Copy all selected objects to the clipboard.

To Launch Ctrl Insert - Alt E C

Purpose The Edit Copy command is used to copy the objects that are currently selected in the workspace to the (internal) Schematic Library Editor clipboard (not the standard Windows clipboard). The Edit Paste command can be used to place a copy of the selection back into any open component library document window.

Outcome Makes a copy of the selection in the clipboard. The clipboard holds the last selection only, each time you use the Copy or Cut command, you overwrite the previous selection.

Comments The Clipboard can be used to paste the selection to other Windows applications that support .WMF (Windows MetaFile) format.

Procedure To copy the current selection from the active window:

1. Make sure that the current selection includes only those objects you wish to copy;

You can use Edit De-Select to de-select objects that are not to be copied and then use Edit Select to select the objects that will be copied. You can also use the shortcut SHIFT+LEFT MOUSE to add objects to the current selection or to de-select any selected objects.

2. Choose Copy from the Edit menu. (shortcut: CTRL+INSERT);

You will be prompted to select a reference point if the Clipboard Reference option is enabled in the Preferences dialog box. A reference point is a coordinate relative to the selected object(s). If the clipboard reference option is off then the current cursor position is used as the reference point. When you paste the selection, the reference point will locate the cursor at this same relative position, allowing you to accurately position the selection.

See also

Paste
Cut

Paste

Summary Place clipboard contents onto current library workspace.

To Launch Shift Insert - Alt E P - 

Purpose The Paste command is used to place the current clipboard contents into any open component library document window. Schematic Library Editor has its own clipboard format. The standard Windows clipboard is not used.

Comments You can repeat the Paste command to duplicate the selection.

Procedure To Paste selected objects from the clipboard to an open document:

1. Choose Paste from the Edit menu. (shortcut: SHIFT+INSERT);

You will be prompted to place the selection and a highlighted outline of the selected objects will be displayed. The cursor position relative to the selection is determined by the Reference Point designated when Cut or Copy was used to add the selection to the clipboard.

2. Position the selection in the library workspace and click LEFT MOUSE or press ENTER.

See also

Cut
Copy

Clear

Summary Delete all selected objects from the current library workspace.

To Launch Ctrl Delete - Alt E L

Purpose The Clear command is used to remove selected objects from the library workspace without copying it to the clipboard.

Outcome The selection will be cleared from the display.

Comments You can use the Edit Undo command (shortcut: ALT+BACKSPACE) to restore the cleared selection.

Procedure To clear the current selection from the active window:

1. Make sure that the current selection includes only those objects you wish to clear.

You can use Edit De-Select to de-select objects that are not to be deleted and then use Edit Select to select the objects that will be deleted. You can also use the shortcut SHIFT+LEFT MOUSE to add objects to the current selection or to de-select any selected objects.

2. Choose Clear from the Edit menu. (shortcut: CTRL+DELETE).

The selection will be cleared from the display.

See also

Undo
Cut
Delete
Delete Single Object

Move Selection

Summary Move selected objects to another area of the library workspace.

To Launch Alt E M S - 

Purpose The Move Selection command is used to reposition an individual selected object or a complex selection containing many objects as a single entity.

Comments While moving a selection, you can rotate it around the cursor and flip it along its x or y axis using the following hot key shortcuts.

spacebar: each press rotates the selection 90 degrees clockwise.

x: flips the selection along its X axis.

y: flips the selection along its Y axis.

Procedure To move selected objects in an open library window:

1. Make sure that the current selection includes only those objects you wish to move.

You can use Edit De-Select to de-select objects that are not to be moved and then use Edit Select to select the objects that will be moved. You can also use the shortcut SHIFT+LEFT MOUSE to add objects to the current selection or to de-select any selected objects.

2. Choose Move and then Move Selection from the Edit menu.

You will be prompted to select a reference point if the Clipboard Reference option is enabled in the Preferences dialog box. A reference point is a coordinate relative to the selected object(s). If the clipboard reference option is off then the current cursor position is used as the reference point. When you move the selection, the reference point will locate the cursor at this same relative position, allowing you to accurately position the selection.

4. Position the selection in the library workspace and click LEFT MOUSE or press ENTER.

See also

Move

Select Inside Area

Summary Select all objects inside an area.

To Launch Alt E S I - 

Purpose The Select Inside Area command is used to select all objects that lie completely within a user-defined rectangle in an open library document window.

Outcome The newly selected objects will be highlighted using the selection color defined in the Options Preferences dialog box. Any previously selected objects will remain selected.

Procedure To select objects within an area:

1. Choose Select and then Inside Area from the Edit menu.

You will be prompted: "Select First Corner."

2. Move the cursor and then press ENTER or LEFT MOUSE to define the first corner of the selection rectangle.

The prompt changes to: "Select Second Corner."

3. Move the cursor to enclose the area to be selected in the highlighted rectangle.

4. Press ENTER or LEFT MOUSE to complete the selection.

See also

Select Outside Area
Select All

Select Outside Area

Summary Select all objects outside an area.

To Launch Alt E S O

Purpose The Select Outside Area command is used to select all objects that lie completely outside a user-defined rectangle in an open library document window.

Outcome The newly selected objects will be highlighted using the selection color defined in the Options Preferences dialog box. Any previously selected objects will remain selected.

Procedure To select objects outside an area:

1. Choose Select and then Outside Area from the Edit menu.

You will be prompted: "Select First Corner."

2. Move the cursor and then press ENTER or LEFT MOUSE to define the first corner of the selection rectangle.

You will then be prompted: "Select Second Corner."

3. Move the cursor to enclose the area to remain de-selected with the highlighted rectangle.

4. Press ENTER or s LEFT MOUSE to complete the selection.

See also

Select Inside Area
Select All

Select All

Summary Select everything on the current library workspace.

To Launch Alt E S A

Purpose The Select All command is used to select all of the objects on the current library workspace.

Comments The selected objects will be highlighted using the selection color defined in the Options Preferences dialog box.

Procedure To select all objects from an open document:

1. Choose Select and then All from the Edit menu.

See also

Select Inside Area
Select Outside Area

De-Select Inside Area

Summary De-Select all objects inside an area.

To Launch Alt E E I

Purpose The De-Select All command is used to de-select all of the objects on the current library workspace.

Procedure To de-select all objects from an open document:

1. Choose De-Select and then All from the Edit menu.

See also

De-Select Outside Area
De-Select All

De-Select Outside Area

Summary De-Select all objects outside an area.

To Launch Alt E E O

Purpose The De-Select Outside Area command is used to de-select all objects that lie completely outside a user-defined rectangle in an open library document window.

Procedure To de-select objects outside an area:

1. Choose De-Select and then Outside Area from the Edit menu.

You will be prompted: "Select First Corner."

2. Move the cursor and then press ENTER or LEFT MOUSE to define the first corner of the selection rectangle.

You will then be prompted: "Select Second Corner."

3. Move the cursor to enclose the area to remain selected with the highlighted rectangle.

4. Press s ENTER or LEFT MOUSE to complete the selection.

See also

Select Outside Area
De-Select Inside Area

De-Select All

Summary De-Select all selected objects.

To Launch Alt E E A - 

Purpose The De-Select All command is used to de-select all of the objects that are currently selected in the library document window.

Procedure To de-select all objects from an open library document window:

1. Choose De-Select and then All from the Edit menu.

See also

De-Select Inside Area
De-Select Outside Area

Toggle Selection

Summary Toggle selection-state of objects.

To Launch Alt E N

Purpose The Toggle Selection command is used to select and de-select objects on a library workspace by moving the cursor over the object and clicking LEFT MOUSE or ENTER.

Outcome Objects that are selected will be outlined in the selection color defined in the Options Preferences dialog box.

Comments The Toggle Selection command is especially useful when working on a densely populated area of a workspace or when you wish to quickly add and remove a number of objects.

Procedure To toggle the selection of any object in an open library document window:

1. Choose Toggle Selection from the Edit menu.

2. Select or de-select the objects in the library document window by moving the cursor over the object and pressing ENTER or LEFT MOUSE.

3. Press ESC or RIGHT MOUSE to leave this command.

See also

Toggle Single Object

Toggle Single Object

Summary Toggle selection-state of objects.

Purpose The Toggle Single object command is used to select and de-select objects in the library document window by moving the cursor over the object and clicking SHIFT + LEFT MOUSE.

Outcome Objects that are selected will be outlined in the selection color.

Procedure To toggle the selection of a single object in an open library document window:

1. Move the cursor over the object to be selected.

2. Press SHIFT + LEFT MOUSE to select or de-select the object.

See also

Toggle Selection

Delete

Summary Select and delete objects on library workspace.

To Launch Alt E D

Purpose The Delete command is used to delete objects in the library document window.

Comments All deletions can be restored by using the Edit Undo command (ALT+BACKSPACE). If you have deleted a series of objects, they will be restored individually, starting with the last object deleted.

Accidental deletion of objects may be further safe-guarded by enabling the Question Delete option. If this option is enabled a Confirm Delete dialog box will be displayed as you select an object. You can enable or disable this message from the Preferences dialog box in the Options menu.

Procedure To Delete objects in an open library document window:

1. Choose Delete from the Edit menu.

2. Delete the objects from the workspace by moving the cursor over an object and pressing ENTER or LEFT MOUSE.

As you delete an object, it will be cleared from the workspace and the status line will continue to prompt you to choose another object.

4. Press ESC or RIGHT MOUSE to leave this command.

See also

Undo
Delete Single Object

Delete Single Object

Summary Delete object that currently has the focus.

Purpose The Delete Single object command is used to delete an object that currently has the focus.

Comments All deletions can be restored by using the Edit Undo command (ALT+BACKSPACE). If you have deleted a series of objects, they will be restored individually, starting with the last object deleted.

Accidental deletion of objects may be further safe-guarded by enabling the Question Delete option. If this option is enabled a Confirm Delete dialog box will be displayed as you select an object. You can enable or disable this message from the Preferences dialog box in the Options menu.

Procedure To delete an object that currently has the focus;

1. Move the cursor over the object to be deleted and click LEFT MOUSE. The object will be displayed with graphical editing handles.

2. Press the DELETE key.

When you press DELETE, the object will be cleared from the library workspace.

See also

Undo
Delete
Clear

Change

Summary Select and use dialog to change objects.

To Launch Alt E H

Purpose The Change command is used to modify specific attributes of placed objects on the library workspace. Each object or object has its own range of editable attributes. You can change one object or extend changes across your entire objects in the workspace using powerful global editing options.

Procedure To change any placed object:

1. Choose Change from the Edit menu.

2. Position the cursor over the target object and press ENTER or LEFT MOUSE.

A dialog box opens, displaying the editable attributes for the object. It is possible to change any or all attributes of the edited object.

3. Press ENTER or click OK to accept your changes. To cancel the change and close the dialog box press ESC or click CANCEL.

4. Choose another object or press ESC or LEFT MOUSE to leave the Change command.

See also

Change Single Object

Change Single Object

Summary Use dialog to change object under the cursor.

Purpose The Change Single Object command is used to modify specific attributes of placed objects on the library workspace. This command is similar to the Change Command, except that when an object has been edited you will not be prompted to select another object. This command is activated by using the shortcut of double clicking the LEFT MOUSE on the object to be changed.

Procedure To change any placed object:

1. Position the cursor over the target object and Double Click the LEFT MOUSE.

A dialog box opens displaying the editable attributes for the object. It is possible to change any or all attributes of the edited object.

2. Press ENTER or click OK to accept your changes. To cancel the changes and close the dialog box press ESC or click CANCEL.

See also

Change

Move

Summary Select and move objects on the library workspace.

To Launch Alt E M M

Purpose The Move command is used to interactively change the location of placed objects on the library workspace.

Comments Schematic Library Editor automatically stacks layers of objects, text and graphics. Each object is on a different layer depending on the creation order. The objects created or added recently are always on the top layer. When you move and place an object it retains its original order in the stack.

You can also move objects by pressing TAB and typing new coordinates directly into the Change dialog box.

Procedure To move any placed object:

1. Choose Move, and then Move from the Edit menu.

2. Position the cursor over the target object and press ENTER or LEFT MOUSE.

If the object does not move with the cursor you may not be exactly over the object when you tried to move it. Use the Zoom options (or press PGUP) to enlarge your view of the workspace and try again.

3. Move the object to the desired location and press ENTER or LEFT MOUSE. To cancel the move press ESC or click RIGHT MOUSE.

4. Move another object or press ESC or LEFT MOUSE to leave the move command.

See also

Move Single Object

Move Single Object

Summary Select and move objects on the library workspace.

Purpose The Move Single Object command is used to interactively change the location of placed objects on the library workspace. This command is similar to the Move command, except that when an object has been moved you will not be prompted to move another object. This command is activated by using the default shortcut: CTRL + LEFT MOUSE.

Comments Schematic Library Editor automatically stacks layers of objects, text and graphics. Each object is on a different layer depending on the creation order. The objects created or added recently are always on the top layer. When you move an object by using the default shortcut of CTRL + LEFT MOUSE and then place the object, it retains its original order in the stack.

While moving the object, you can rotate it 90 degrees around the cursor by pressing the SPACEBAR and flip it along its x or y axis by using the X and Y keys. Press TAB to manually change the location of the object by typing new coordinates directly into a dialog box.

The Move Single Object command can only be accessed using the mouse, unless its hot key is changed in the Assign Hot Keys dialog box.

Procedure To move any placed object:

1. Position the cursor over the target object and press CTRL + LEFT MOUSE.

2. Move the object to the desired location and press ENTER or LEFT MOUSE.

If the object does not move with the cursor you may not be exactly over the object when you tried to move it. Use the Zoom options (or press PGUP) to enlarge your view of the workspace and try again.

3. To cancel the move press ESC or click RIGHT MOUSE.

See also

Move

Move To Front

Summary Move and place an object in the front of all other objects.

To Launch Alt E M V

Purpose The Move To Front command is used to interactively change the location of placed objects on the library workspace. This command is similar to the Move command, except that when an object has been moved and placed on the workspace it will be positioned in front of all the other objects on the workspace.

Comments Schematic Library Editor automatically stacks layers of objects, text and graphics. Each object is on a different layer depending on the creation order. The objects created or added recently are always on the top layer. When you use the Move To Front command and place an object it will be positioned at the top of the stack in front of all the other objects on the workspace.

While moving the object, you can rotate it 90 degrees around the cursor by pressing the SPACEBAR and flip it along its x or y axis by using the X and Y keys. Press TAB to manually change the location of the object by typing new coordinates directly into a dialog box.

Procedure To move any placed object to the front of all other objects on the workspace:

1. Choose Move and then Move To Front from the Edit menu.

2. Position the cursor over the target object and press ENTER or LEFT MOUSE.

If the object does not move with the cursor you may not be exactly over the object when you tried to move it. Use the Zoom options (or press PGUP) to enlarge your view of the workspace and try again.

3. Move the object to the desired location and press ENTER or LEFT MOUSE. To cancel the move press ESC or click RIGHT MOUSE.

4. Move another object or press ESC or RIGHT MOUSE to leave the Move To Front command.

See also

Bring To Front

Bring To Front

Summary Bring an object graphically to the front of all other objects.

To Launch Alt E M F

Purpose The Bring To Front command is used to move objects on the library workspace to the front of all other objects.

Comments Schematic Library Editor automatically stacks layers of objects, text and graphics. Each object is on a different layer depending on the creation order. The objects created or added recently are always on the top layer. When you use the Bring To Front command and click on an object it will be positioned at the top of the stack in front of all the other objects on the workspace.

Procedure To move any placed objects to the front of all other objects on the workspace:

1. Choose Move and then Bring To Front from the Edit menu.

2. Position the cursor over the target object and press ENTER or LEFT MOUSE.

When the object has been chosen it will be positioned in front of all the other objects on the workspace.

3. Click on another object or press ESC or RIGHT MOUSE to leave the Bring To Front command.

See also

Move To Front

Send To Back

Summary Send an object to the back of all other objects.

To Launch Alt E M B

Purpose The Send To Back command is used to send objects on the library workspace to the back of all other objects.

Comments Schematic Library Editor automatically stacks layers of objects, text and graphics. Each object is on a different layer depending on the creation order. The objects created or added recently are always on the top layer. When you use the Send To Back command and click on an object it will be positioned at the bottom of the stack, behind all the other objects on the workspace.

Procedure To send an object to the back of all other objects on the workspace:

1. Choose Move and then Send To Back from the Edit menu.

2. Position the cursor over the target object and press ENTER or LEFT MOUSE.

When the object has been chosen it will be positioned at the back of all the other objects on the workspace.

3. Click on another object or press ESC or RIGHT MOUSE to leave the Send To Back command.

See also

Send To Back Of

Bring To Front Of

Summary Bring an object to the front of another object.

To Launch Alt E M O

Purpose The Bring To Front Of command is used to move an object on the library workspace to the front of another object. This command is similar to the Bring To Front command, except that you will be prompted to choose the object that the original object will be placed in front of.

Comments Schematic Library Editor automatically stacks layers of objects, text and graphics. Each object is on a different layer depending on the creation order. The objects created or added recently are always on the top layer. When you use the Bring To Front Of command the original object will be placed in front of the target object.

Procedure To move a placed object to the front of another object on the workspace:

1. Choose Move and then Bring To Front Of from the Edit menu.

2. Position the cursor over the target object and press ENTER or LEFT MOUSE.

The object will temporarily disappear from the workspace exposing any objects behind it.

3. Click on the object that the original object will be placed in front of.

4. Click on another object or press ESC or RIGHT MOUSE to leave the Bring To Front Of command.

See also

Bring To Front

Send To Back Of

Summary Send an object to the back of another object.

To Launch Alt E M T

Purpose The Send To Back Of command is used to move an object on the library workspace behind another object. This command is similar to the Send To Back command, except that you will be prompted to select the object that the original object will be placed behind.

Comments Schematic Library Editor automatically stacks layers of objects, text and graphics. Each object is on a different layer depending on the creation order. The objects created or added recently are always on the top layer. When you use the Send To Back Of command the original object will be placed behind the target object.

Procedure To move a placed object behind another object on the workspace:

1. Choose Move and then Send To Back Of from the Edit menu.

2. Position the cursor over the target object and press ENTER or LEFT MOUSE.

The object will temporarily disappear from the workspace exposing any objects behind it.

3. Click on the object that the original object will be placed behind.

Press ESC or RIGHT MOUSE to leave the Send To Back Of command.

See also

Send To Back

Place Arc

Summary Place graphical arcs on the library workspace.

To Launch Alt P A

Purpose The Place Arc command is used to place an arc on the library workspace using the center of the arc as the starting point. Arcs are used for building library components, such as defining component outlines or adding graphical symbols.

Comments When using the Place Arc command the default placement field values are determined by the last placed arc. Press TAB to change the default arc line width and color during placement.

Procedure To place an arc on the library workspace:

1. Choose Arcs from the Place menu.

While moving the arc, you can rotate it 90 degrees around the cursor and flip it along its x or y axis by using SPACEBAR, x and y keys. Press TAB to manually change the default values of the arc by typing new values directly into a dialog box.

2. Position the cursor where the center of the arc is to be placed and press ENTER or LEFT MOUSE once.

As you move the mouse or press the arrow keys, a highlighted arc will be displayed.

3. Position the cursor to establish the desired radius then press ENTER or LEFT MOUSE.

4. Position the cursor to define the start angle of the arc, then press ENTER or LEFT MOUSE again.

5. Position the cursor to define the end angle of the arc, then press ENTER or LEFT MOUSE.

Start a new arc by selecting a center point or press ESC or RIGHT MOUSE to exit this command.

See also

Arc
Place Elliptical Arc

Place Polygon

Summary Place graphical polygon shapes on the library workspace.

To Launch Alt P Y - 

Purpose The Place Polygon command is used to place graphical polygon objects on the current library workspace. Polygons are used for building library components, such as defining the component body or adding graphical symbols.

Comments When using the Place Polygon command the default placement field values are determined by the last placed polygon. Press TAB to change the default polygon border width, border color and fill color during placement.

Procedure To place a polygon in the library workspace:

1. Choose Polygons from the Place menu.

Press TAB to manually change the default values of the polygon by typing new values directly into a dialog box.

2. Position the cursor where the first polygon segment is to be placed and press ENTER or LEFT MOUSE once.

As you move the mouse or press the arrow keys, a highlighted polygon outline will be displayed.

You may continue to add polygon line segments as desired by clicking the LEFT MOUSE or pressing the INSERT key. If you make a mistake, you can press DELETE to remove the last line segment. You can also press ESC or RIGHT MOUSE to cancel the current segment.

5. To exit this command press ESC or RIGHT MOUSE a second time.

See also

Polygon

Place Line

Summary Place graphical lines on the library workspace.

To Launch Alt P L - 

Purpose The Place Polyline command is used to place graphical lines on the current library workspace. Polylines are used for building library components, such as defining component outlines or adding graphical symbols.

Comments When using the Place Line command the default placement field values are determined by the last placed line. Press TAB to change the default line line width, line style and color during placement.

Procedure To place a line on the library workspace:

1. Choose line from the Place menu.

Press TAB to manually change the default values of the polyline by typing new values directly into a dialog box.

2. Position the cursor where the first polyline segment is to be placed and press ENTER or LEFT MOUSE once.

As you move the mouse or press the arrow keys, a highlighted polyline segment will be displayed.

3. Position the cursor to establish the length and angle of the polyline segment, then press ENTER or LEFT MOUSE to end this first segment of the line.

You may continue to add line segments as desired by clicking the LEFT MOUSE or pressing the INSERT key. If you make a mistake, you can press DELETE to remove the last line segment. You can also press ESC or RIGHT MOUSE to cancel the current segment.

5. To exit this command press ESC or s RIGHT MOUSE a second time.

See also

Line

Place Bezier

Summary Place curved lines on the library workspace.

To Launch Alt P B - 

Purpose The Place Bezier command is used to place curved line segments on the current library workspace. Beziers are used for building library components, such as defining component outlines or adding graphical symbols.

Comments When using the Place Bezier command the default placement field values are determined by the last placed Bezier. Press TAB to change the default Bezier curve width and color during placement.

Procedure To place a Bezier on the library workspace:

1. Choose Beziers from the Place menu.

Press TAB to manually change the default values of the Bezier by typing new values directly into a dialog box.

2. Position the cursor where the first Bezier vertex is to be placed and press ENTER or LEFT MOUSE once. A Bezier must have at least four vertices to form a curve. The maximum number of vertices supported by a single Bezier object is fifty.

As you move the mouse or press the arrow keys, a highlighted Bezier will be displayed.

You may continue to add Bezier vertices as desired by clicking the LEFT MOUSE or pressing the INSERT key. If you make a mistake, you can press DELETE to remove the last line segment. You can also press ESC or RIGHT MOUSE to cancel the current segment.

5. To exit this command press ESC or RIGHT MOUSE a second time.

See also

Bezier

Place Ellipse

Summary Place elliptical shapes on the library workspace.

To Launch Alt P E - 

Purpose The Place Ellipse command is used to place elliptical shapes on the library workspace using the center of the ellipse as the starting point. Ellipses are used for building library components, such as defining the component body or adding graphical symbols.

Comments When using the Place Ellipse command the default placement field values are determined by the last placed ellipse. Press TAB to change the default ellipse border width, border color and fill color during placement.

Procedure To place an ellipse on the library workspace:

1. Choose Ellipses from the Place menu.

While moving the ellipse, you can rotate it 90 degrees around the cursor and flip it along its x or y axis by using the spacebar, x and y keys. Press TAB to manually change the default values of the ellipse by typing new values directly into a dialog box.

2. Position the cursor where the center of the ellipse is to be placed and press ENTER or LEFT MOUSE once.

As you move the mouse or press the arrow keys, a highlighted ellipse will be displayed.

3. Position the cursor to establish the desired x-radius then press ENTER or LEFT MOUSE.

4. Position the cursor to establish the desired y-radius then press ENTER or LEFT MOUSE.

Start a new ellipse by selecting a center point or press ESC or RIGHT MOUSE to exit this command.

See also

Ellipse
Place Elliptical Arc

Place Text

Summary Place single line text on the library workspace.

To Launch Alt P T - 

Purpose The Place Text command is used to place single line notes or descriptive text around a library component.

Comments When using the Place Text command the default placement field values are determined by the last placed text string. Press TAB to change the default text, font style, color and orientation during placement.

Procedure To place single line text on the library workspace:

1. Choose Text from the Place menu.

While moving the text, you can rotate it 90 degrees around the cursor and flip it along its x or y axis by using the spacebar, x and y keys. Press TAB to manually change the default values of the text by typing new values directly into a dialog box.

2. Position the cursor where the text is to be placed and press ENTER or LEFT MOUSE once.

Place another text string or press ESC or RIGHT MOUSE to exit this command.

See also

Text

Place Pie Charts

Summary Place pie charts on the library workspace.

To Launch Alt P C

Purpose The Place Pie command is used to place circular graphical objects on the workspace. Pie charts are used for building library components, such as defining the component body or adding graphical symbols.

Comments When using the Place Pie command the default placement field values are determined by the last placed pie chart. Press TAB to change the default pie chart border width, border color and fill color during placement.

Procedure To place a pie chart on the library workspace:

1. Choose Pie Charts from the Place menu.

While moving the pie chart, you can rotate it 90 degrees around the cursor and flip it along its x or y axis by using the spacebar, x and y keys. Press TAB to manually change the default values of the pie chart by typing new values directly into a dialog box.

2. Position the cursor where the center of the pie chart is to be placed and press ENTER or LEFT MOUSE once.

As you move the mouse or press the arrow keys, a highlighted pie chart will be displayed.

3. Position the cursor to establish the desired radius then press ENTER or LEFT MOUSE.

4. Position the cursor to define the start angle of the pie chart, then press ENTER or LEFT MOUSE again.

5. Position the cursor to define the end angle of the pie chart, then press ENTER or LEFT MOUSE.

Place another pie chart or press ESC or RIGHT MOUSE to exit this command.

See also

Pie Chart

Place Pin

Summary Place electrical pin on the library workspace.

To Launch Alt P P - 

Purpose The Place Pin command is used to place Part pins on the workspace. Pins are used to direct signals into and out of Parts. Pins have a user defined name and number, electrical type, length, color, orientation and other display options.

Comments While placing pins on the workspace the pin number will automatically increment if the first or last characters are numbers. For example, an initial pin number of 1 will automatically increment to 2, 3, 4... and so on. A number of 2A will increment 3A, 4A, 5A, and a number of A10 will increment A11, A12, A13.

When using the Place Pin command the default placement field values are determined by the last placed Pin. Press TAB to change the default symbol type and color during placement.

Procedure To place pins on the library workspace:

1. Choose Pins from the Place menu.

While moving the Pin, you can rotate it 90 degrees around the cursor by pressing the spacebar. Press TAB to manually change the default values of the pin by typing new values directly into a dialog box.

As you move the mouse or press the arrow keys, a highlighted pin will be displayed.

2. Position the cursor where the pin is to be placed and press ENTER or LEFT MOUSE once.

Place another pin or press ESC or RIGHT MOUSE to exit this command.

See also

Pin

Place Round Rectangle

Summary Place round rectangles on the library workspace.

To Launch Alt P O - 

Purpose The Place Round Rectangle command is used to place a rounded rectangle on the library workspace using the center of the rounded rectangle as the starting point. Rounded rectangles are used for building library components, such as defining the component body or adding graphical symbols.

Comments When using the Place Round Rectangle command the default placement field values are determined by the last placed round rectangle. Press TAB to change the default round rectangle border width, border color and fill color during placement.

Procedure To place a rounded rectangle on the library workspace:

1. Choose Round Rectangle from the Place menu.

While moving the rounded rectangle, you can rotate it 90 degrees around the cursor and flip it along its x or y axis by using the spacebar, x and y keys. Press TAB to manually change the default values of the rounded rectangle by typing new values directly into a dialog box.

2. Position the cursor where the center of the rounded rectangle is to be placed and press ENTER or LEFT MOUSE once.

As you move the mouse or press the arrow keys, a highlighted rounded rectangle will be displayed.

3. Position the cursor to establish the desired shape, then press ENTER or LEFT MOUSE.

Place another rounded rectangle or press ESC or RIGHT MOUSE to exit this command.

See also

Round Rectangle
Place Rectangle

Place Elliptical Arc

Summary Place elliptical arcs on the library workspace.

To Launch Alt P I - 

Purpose The Place Elliptical Arc command is used to place an elliptical arc on the library workspace using the center of the elliptical arc as the starting point. Elliptical arcs are used for building library components, such as defining component outlines or adding graphical symbols.

Comments When using the Place Elliptical Arc command the default placement field values are determined by the last placed elliptical arc. Press TAB to change the default arc line width and color during placement.

Procedure To place an elliptical arc on the library workspace:

1. Choose Elliptical Arcs from the Place menu.

While moving the elliptical arc, you can rotate it 90 degrees around the cursor and flip it along its x or y axis by using the spacebar, x and y keys. Press TAB to manually change the default values of the elliptical arc by typing new values directly into a dialog box.

2. Position the cursor where the center of the elliptical arc is to be placed and press ENTER or LEFT MOUSE once.

As you move the mouse or press the arrow keys, a highlighted arc will be displayed.

3. Position the cursor to establish the desired x-radius then press ENTER or LEFT MOUSE.

4. Position the cursor to establish the desired y-radius then press ENTER or s LEFT MOUSE.

5. Position the cursor to define the start angle of the elliptical arc, then press ENTER or LEFT MOUSE again.

6. Position the cursor to define the end angle of the elliptical arc, then press ENTER or LEFT MOUSE.

Start a new elliptical arc by selecting a center point or press ESC or RIGHT MOUSE to exit this command.

See also

Elliptical Arc
Place Arc

Place Rectangle

Summary Place rectangles on the library workspace.

To Launch Alt P R - 

Purpose The Place Rectangle command is used to place a graphical rectangle on the library workspace. Rectangles are used for building library components, such as defining the component body or adding graphical symbols.

Comments When using the Place Rectangle command the default placement field values are determined by the last placed rectangle. Press TAB to change the default rectangle border width, border color and fill color during placement.

Procedure To place a rectangle on the library workspace:

1. Choose Rectangle from the Place menu.

While moving the rectangle, you can rotate it 90 degrees around the cursor and flip it along its x or y axis by using the spacebar, x and y keys. Press TAB to manually change the default values of the rectangle by typing new values directly into a dialog box.

2. Position the cursor where the center of the rectangle is to be placed and press ENTER or LEFT MOUSE once.

As you move the mouse or press the arrow keys, a highlighted rectangle will be displayed.

3. Position the cursor to establish the desired shape, then press ENTER or LEFT MOUSE.

Place another rectangle or press ESC or RIGHT MOUSE to exit this command.

See also

Rectangle
Place Round Rectangle

Place IEEE Symbol

Summary Place IEEE Symbols on the library workspace.

Purpose The Place IEEE Symbol command is used to place graphical qualifying symbols on the workspace. IEEE symbols are used for representing logic functions or devices. These symbols enable users to understand the logic characteristics of these functions or devices without requiring specific knowledge of their internal characteristics.

Comments When using the Place IEEE Symbol command the default placement field values are determined by the last placed IEEE symbol. Press TAB to change the default symbol type and color during placement.

Procedure To place an IEEE symbol on the library workspace:

1. Choose IEEE Symbols from the Place menu.

While moving the IEEE symbol, you can rotate it 90 degrees around the cursor and flip it along its x or y axis by using the spacebar, x and y keys. Press TAB to manually change the default values of the IEEE symbol by typing new values directly into a dialog box.

2. Position the cursor where the center of the IEEE symbol is to be placed and press ENTER or LEFT MOUSE once.

As you move the mouse or press the arrow keys, a highlighted IEEE symbol will be displayed.

3. Position the cursor where the IEEE symbol is to be placed and press ENTER or LEFT MOUSE once.

Place another IEEE symbol or press ESC or RIGHT MOUSE to exit this command.

See also

IEEE Symbol

Place Junction Symbol

Summary Place a Junction symbol on the library workspace.

To Launch Alt P J - 

Purpose The Place Junction Symbol command is used to place a graphical circular object on the library workspace. Junction symbols are used for building library components.

Comments Junction symbols are graphical only and have no logical connectivity.

Procedure To place a junction symbol on the library workspace:

1. Choose Junction Symbol from the Edit menu.

   Press TAB to manually change the default values of the junction by typing new values directly into a dialog box.

2. Position the cursor where the center of the junction is to be placed and press ENTER or LEFT MOUSE once.

   Place another Junction or press ESC or RIGHT MOUSE to exit this command.

See also

Junction Symbol

Place Graphic

Summary Place PCX/BMP/GIF/TIFF/WMF/EPS graphical images on the library workspace.

To Launch Alt P G - 

Purpose The Place Graphic command is used for adding reference information to a library component, such as mechanical and electrical symbols or customized component images.

Comments Images are linked during loading, they are not stored in the library. Only the image file path is stored in the library.

Schematic Library Editor supports the import of the following image formats;

BMP - All uncompressed Bitmap images. Windows device-independent bitmap format, introduced with Windows 3.0 and increasingly supported by Windows applications.

PCX - Paintbrush format, used in Windows Paintbrush and other paint programs and supported by many desktop publishing and graphics programs. Supported colors include, monochrome, 16 color, 256 color, 24-bit color.

TIFF - Tag Image File Format, supported by many desktop publishing programs. Supported compression types, uncompressed, LZW, Packbits, Modified Huffman encoding, CCITT Group 3 1D, CCITT Group 3 2D, CCITT Group 4. Supported colors include, monochrome, 16 color, 256 color, 24-bit color

GIF - All non-interlaced Graphic Image files.

EPSF - Encapsulated postscript files with and without display images. If the EPS file does not contain a TIFF or Windows Metafile display image then the file name of the EPS image will be displayed.

WMF - Only Windows Metafiles which conform to the Aldus Placeable Metafile Format are supported. Most applications which export or import Metafiles support this format.

Procedure To place graphical objects on the library workspace;

1. Choose Graphic from the Place menu.

2. Type the image file name (include the full path, if different from the path listed after Directory) and click OK or press ENTER.

You can also double-click on the desired file name in the Files window, if any. To change directories, click under any of the options listed in the Directories window.

Press TAB to manually change the default values of the graphic image by typing new values directly into a dialog box.

3. Position the cursor where the center of the graphic image is to be placed and press ENTER or LEFT MOUSE once.

As you move the mouse or press the arrow keys, a highlighted rectangle will be displayed.

4. Position the cursor to establish the desired size of the image, then press ENTER or

LEFT MOUSE.

5. The Image file dialog box will be displayed again, choose another graphic image or press ESC or click Cancel to exit this command.

See also

Graphic

Paste Array

Summary Define an array placement of clipboard contents.

To Launch Alt E Y - 

Purpose The Paste Array command is used to define a multiple placement of the clipboard contents.

Outcome The following are options that can be set within the Setup Array Placement dialog box.

Item Count - The number of repeat placements to be performed. For example, typing 5 will place 5 of the current selection, contained in the clipboard.

Text Increment - This option is used for pin names and annotations. Setting this to 1 will automatically increment. For example, an initial Pin number of A1 will automatically increment to A2, A3, A4 and so on, and a Pin number of 1A will increment 2A, 3A, 4A.

Incrementing can be numeric (1,2,3) or alphabetic (A, B, C) or a combination of alpha and numeric (A1, A2, or 1A, 1B or 1A, 2A, etc.).

Horizontal - Specifies the distance along the x-axis between each clipboard selection as it is placed. Positive and negative values are supported.

Vertical - Specifies the distance along the y-axis between each clipboard selection as it is placed. Positive and negative values are supported.

Comments The clipboard holds the last selection only, each time you use the Cut or Copy command, you overwrite the previous selection.

Procedure To setup an array placement:

1. Choose Paste Array from the Edit menu.

The Setup Array Placement dialog box appears.

2. Type in new array placement settings.

3. Press the OK button to accept the array definition and return to the editor without executing the placement or;

4. Press the Place button to execute the array placement with the current clipboard contents. Position the cursor at the desired reference point for the array placement and click ENTER or LEFT MOUSE.

See also

Cut
Copy
Execute Paste Array

Execute Paste Array

Summary Place array with the last set array placement options.

Purpose The Execute Paste Array command is used to paste an array using the current clipboard contents and the last array setup options. This command is accessed using the Place button on the Setup Array Placement dialog box.

Outcome The clipboard contents will be placed according to the array setup you have defined, starting at the reference point you indicated.

Comments The Execute Paste Array command may be assigned a hot key in the Assign Hot Keys dialog box to allow quick array placements.

Procedure To execute an array placement using the current clipboard contents

1. Choose Paste Array from the Place menu.

The Setup Array Placement dialog box appears.

2. Press the Place button to execute the array placement with the current clipboard contents.

3. Position the cursor at the desired reference point for the array placement and click ENTER or LEFT MOUSE.

See also

Paste Array
Hot Keys

Jump To Origin

Summary Jump to the origin of the library workspace (lower left).

To Launch Alt E J O

Purpose The Jump Origin command is used to move the cursor to the center of the library workspace. This can save you from having to constantly zoom in and out to navigate around your design and is particularly useful for large or complex components.

Outcome Once the Jump To Origin command has been selected the cursor will jump to the lower left corner (0,0 coordinate) of the library workspace.

Procedure To move the cursor to the origin of the library workspace:

1. Choose Jump To Origin from the Edit menu.

Zoom Window

Summary Select a rectangular area of the workspace and fit that area in the window.

To Launch Alt Z W

Purpose The Zoom Window command is used to re-set the magnification of a selected area in the current document window.

Procedure To magnify a selected area in an open document:

1. Choose Window from the Zoom menu.

2. Move the cursor and then press ENTER or LEFT MOUSE to define the first corner of the zoom rectangle.

3. Move the cursor to enclose the area to be magnified with the highlighted rectangle.

4. Press ENTER or LEFT MOUSE to complete zoom window.

See also

Zoom Point
Zoom In
Zoom Out

Zoom Point

Summary Select a rectangular area of the workspace and fit that area in the window.

To Launch Alt Z P

Purpose The Zoom Point command is used to re-set the magnification from a selected point and defined area in the current document window.

Procedure To magnify a selected area in an open document:

1. Choose Point from the Zoom menu.

You will be prompted: "Select First Corner."

2. Move the cursor and then press ENTER or LEFT MOUSE to define the starting point the zoom rectangle.

You will then be prompted: "Select Second Corner."

3. Move the cursor to enclose the area to be magnified with the highlighted rectangle.

4. Press ENTER or LEFT MOUSE to complete the zoom window.

See also

Zoom Window
Zoom In
Zoom Out

Zoom Workspace

Summary Show the entire workspace.

To Launch Alt Z S

Purpose The Zoom Workspace command is used to view the entire area of the current workspace.

Procedure To view all of the current workspace:

1. Choose Workspace from the Zoom menu.

See also

Zoom All

Zoom All

Summary Fit all objects on the current workspace in the window.

To Launch Ctrl PgDn - Alt Z A

Purpose The Zoom All command is used to fit all of the objects on the current library workspace into the current document window.

Comments The zoom window region is defined by objects at the highest and lowest coordinate points.

Procedure To fit all objects on the current workspace into the current document window:

1. Choose All from the Zoom menu.

See also

Zoom Workspace

Zoom In

Summary Show less of the current workspace (higher magnification).

To Launch PgUp - Alt Z I - 

Purpose The Zoom In command is used to increase the magnification of the current workspace.

Comments The Zoom In command is most effective when initiated with its shortcut: PGUP or Z, I.

Procedure To increase the magnification of the current workspace:

1. Press PGUP or Z, I.

See also

Zoom Out
Zoom All

Zoom Out

Summary Show more of the current workspace (lower magnification).

To Launch PgDn - Alt Z O - 

Purpose The Zoom Out command is used to decrease the magnification of the current workspace.

Comments The Zoom Out command is most effective when initiated with its shortcut: PGDN or Z, O.

Procedure To decrease the magnification of the current workspace:

1. Press PGDN or Z, O.

See also

Zoom In
Zoom All

Zoom Pan

Summary Re-Center the screen around the cursor.

To Launch Home - Alt Z N

Purpose The Zoom Pan command is used to redraw the current document window with the previous cursor location at the center of the window.

Comments The Zoom Pan command is most effective when initiated with its shortcut HOME or Z, N.

Procedure To center the workspace view around the current cursor location:

1. Move the cursor to the area or object you wish to position at the center of the current document window.

2. Press HOME or Z, N.

Screen Redraw

Summary Update the screen display.

To Launch End - Alt Z R

Purpose The Zoom Redraw command is used to update or refresh all objects on the current workspace.

Procedure To re-draw the objects on the current workspace:

1. Choose Redraw from the Zoom menu.

Zoom 400%

Summary Set zoom scale to 4x (highest magnification).

To Launch Ctrl 4 - Alt Z 4

Purpose The Zoom 400% command is used to set the magnification of the current workspace to a factor of 4 times the normal scale. This is the highest zoom magnification.

Outcome The view of the current workspace will be re-set to a magnification of 400%, centered around the current cursor location.

Comments The Zoom 400% command is most effective when initiated with its shortcut Z, 4.

Procedure To set the magnification of the current workspace to 400%:

1. Position the cursor on the area to be magnified.

2. Choose 400% from the Zoom menu or press Z, 4.

See also

Zoom All

Zoom 200%

Summary Set zoom scale to 2x.

To Launch Ctrl 2 - Alt Z 2

Purpose The Zoom 200% command is used to set the magnification of the current workspace to a factor of 2 times the normal scale.

Outcome The view of the current workspace will be re-set to a magnification of 200%, centered around the current cursor location.

Comments The Zoom 200% command is most effective when initiated with its shortcut Z, 2.

Procedure To set the magnification of the current workspace to 200%:

1. Position the cursor on the area to be magnified.

2. Choose 200% from the Zoom menu or press Z, 2.

See also

Zoom All

Zoom 100%

Summary Set zoom scale to 1x (normal magnification).

To Launch Ctrl 1 - Alt Z 1

Purpose The Zoom 100% command is used to set the magnification of the current workspace to its normal scale.

Outcome The view of the current workspace will be re-set to a magnification of 100%, centered around the current cursor location.

Comments The Zoom 100% command is most effective when initiated with its shortcut Z, 1.

Procedure To set the magnification of the current workspace to 100%:

1. Position the cursor on the area to be magnified.

2. Choose 100% from the Zoom menu or press Z, 1.

See also

Zoom All

Zoom 50%

Summary Set zoom scale to 0.5x.

To Launch Ctrl 5 - Alt Z 5

Purpose The Zoom 50% command is used to set the magnification of the current workspace to a factor of half the normal scale.

Outcome The view of the current workspace will be re-set to a magnification of 50%, centered around the current cursor location.

Comments The Zoom 50% command is most effective when initiated with its shortcut Z, 5.

Procedure To set the magnification of the current workspace to 50%:

1. Position the cursor on the area to be magnified.

2. Choose 50% from the Zoom menu or press Z, 5.

See also

Zoom All

Report Component

Summary Display component information.

To Launch Alt F R C

Purpose The Report Component command is used to retrieve general information about the current component.

Outcome Once this command is selected the report file is automatically loaded into a text editor and displayed on the screen. The text editor is specified in the Setup Run Options dialog box, (Tools-Setup). Windows Notepad is used by default.

Procedure To get information about the Component on the current workspace:

1. Choose Report from the File menu then choose component.

See also

Report Library

Summary Display library statistics.

To Launch Alt F R L

Purpose The Information Library command is used to retrieve general information about the current library.

Outcome Once this command is selected the report file is automatically loaded into a text editor and displayed on the screen. The text editor is specified in the Setup Run Options dialog box, (Tools-Setup). Windows Notepad is used by default.

Procedure To get information about the current library;

1. Choose Library from the Info menu.

See also

Workspace Options

Summary Setup options for current workspace.

To Launch Alt O W

Purpose The Workspace Options command is used to define various workspace settings that apply to a specific library file. This command is used to define various workspace settings, such as workspace orientation, background color, border color and an option to display or hide the workspace border.

Outcome The following are options that can be set within the Workspace Options dialog box.

Size - Schematic Library Editor supports 10 standard imperial and metric sheet sizes. They include A, B, C, D, E (or metric sizes A4-A0).

Orientation - Defines the orientation of the library workspace. The sheet can be displayed in landscape (default) mode or in portrait mode.

Show Border - If this option is enabled, the sheet border will be displayed. If you want to design a custom border, turn this option off and use Schematic Library Editor's drawing tools.

Custom Size - This is the flag that allows you to use Schematic Library Editor standard workspace sizes or to define your own custom workspace size.

Border Color - The surrounding border of your library workspace can be assigned a default color. To assign a new color to the sheet border, click in the color box to open the Color Selector dialog box. Schematic Library Editor will display all the available colors that your graphics card and monitor supports. The border color only applies to the Schematic Library Editors default border. Customized borders will not be affected by this option.

Workspace Color - A library workspace can be assigned a color. To assign a new color to the workspace, click in the color box to open the Color Selector dialog box.

Snap Grid - If this option is enabled, the visible grid is displayed. The visible grid can be assigned any color and can be displayed as lines or dots.

Visible Grid - If this option is enabled, the visible grid is displayed. The visible grid can be assigned any color and can be displayed as lines or dots.

Procedure To setup and define workspace settings;

1. Choose Workspace from the Options menu

2. Type in new workspace settings.

3. Click OK to accept the new settings.

See also

Preferences

Toggle Scroll Bars

Summary Turn the scroll bars on or off.

To Launch Alt O C

Purpose The Toggle Scroll Bars command is used to turn the Scroll Bars on or off in the current document window. Turning the scroll bars off will display more of the library workspace.

Outcome Once the Scroll Bars command has been chosen the scroll bars at the edge of the current document window will toggle on or off depending on their previous state.

Procedure To toggle the scroll bars on or off:

1. Choose Scroll Bars from the Options menu.

Toggle Visible Grid

Summary Turn the visible grid on or off.

To Launch Alt O V

Purpose The Toggle Visible Grid command is used to turn the visible grid on or off in the current document window. The visible grid provides visual reference as you move around the library workspace and can be displayed as dots or straight lines.

Outcome Once the Visible Grid command has been chosen the visible grid will toggle on or off depending on its previous state.

Comments The visible grid can also be toggled on or of in the Options Workspace dialog box.

Procedure To turn the visible grid on or off:

1. Choose Visible Grid from the Options menu.

See also

Preferences
Workspace Options

Toggle Snap Grid

Summary Turn the snap grid on or off.

To Launch Alt O G

Purpose The Toggle Snap Grid command is used to turn the cursor snap grid on or off. The snap grid defines an array of points in the workspace which restrict cursor movement and the placement of primitives. Turning the snap grid off allows you to place primitives at any location on the workspace without any restrictions.

Outcome Once the Snap Grid command has been chosen the cursor snap grid will toggle on or off depending on its previous state.

Procedure To turn the snap grid on or off:

1. Choose Snap Grid from the Options menu.

Shift Cursor Up

Summary Move the cursor up 10 snap grid points.

Purpose The Shift Cursor Up command is used to move the cursor up 10 grid points in the library workspace.

Outcome When the Shift and Up arrow keys are pressed the cursor will move up 10 grid points, along the X axis.

Procedure To move the cursor up 10 snap grid points:

1. Press the up arrow on the keyboard while holding down the shift key.

See also

Cursor Up

Shift Cursor Down

Summary Move the cursor down 10 snap grid points.

Purpose The Shift Cursor Down command is used to move the cursor down 10 grid points in the library workspace.

Outcome When the Shift and Down arrow keys are pressed the cursor will move down 10 grid points, along the Y axis.

Procedure To move the cursor down 10 snap grid points:

1. Press the down arrow on the keyboard while holding down the shift key.

See also

Cursor Down

Shift Cursor Left

Summary Move the cursor up 10 snap grid points.

Purpose The Shift Cursor Left command is used to move the cursor to the left 10 grid points in the library workspace.

Outcome When the Shift and Left arrow keys are pressed the cursor will move to the left 10 grid points, along the X axis.

Procedure To move the cursor left 10 snap grid points:

1. Press the left arrow key on the keyboard while holding down the shift key.

See also

Cursor Left

Shift Cursor Right

Summary Move the cursor right 10 snap grid points.

Purpose The Shift Cursor Right command is used to move the cursor to the right 10 grid points in the library workspace.

Outcome When the Shift and Right arrow keys are pressed the cursor will move to the right 10 grid points, along the X axis.

Procedure To move the cursor right 10 snap grid points:

1. Press the right arrow on the keyboard while holding down the shift key.

See also

Cursor Right

Cursor Up

Summary Move the cursor up one snap grid point.

Purpose The Cursor Up command is used to move the cursor up one snap grid point in the library workspace.

Outcome When the Up arrow key is pressed the cursor will move up one grid point, along the Y axis.

Procedure To move the cursor up one snap grid point:

1. Press the up arrow on the keyboard.

See also

Shift Cursor Up

Cursor Down

Summary Move the cursor down one snap grid point.

Purpose The Cursor Down command is used to move the cursor down one snap grid point in the library workspace.

Outcome When the Down arrow key is pressed the cursor will move down one grid point, along the Y axis.

Procedure To move the cursor down one snap grid point:

1. Press the down arrow on the keyboard.

See also

Shift Cursor Down

Cursor Left

Summary Move the cursor left one snap grid point.

Purpose The Cursor Left command is used to move the cursor left one snap grid point in the library workspace.

Outcome When the Left arrow key is pressed the cursor will move left one grid point, along the X axis.

Procedure To move the cursor left one snap grid point:

1. Press the left arrow on the keyboard.

See also

Shift Cursor Left

Cursor Right

Summary Move the cursor right one snap grid point.

Purpose The Cursor Right command is used to move the cursor right one snap grid point in the library workspace.

Outcome When the Right arrow key is pressed the cursor will move right one grid point, along the X axis.

Procedure To move the cursor right one snap grid point:

1. Press the up arrow on the keyboard.

See also

Shift Cursor Right

New Part

Summary Add a new part to the currently displayed component.

To Launch Alt C W - 

Purpose The New Part command is used to create a new part within the current component.

Outcome When you choose the New Part command a new part with no primitives is added to the currently displayed component.

The Document Title bar displays the component name and the number of the part, for example, Component_1 2/2.

Comments Use the Copy and Paste commands from the Edit menu to copy the contents of a previously created part into the current part.

Schematic Library Editor allows an unlimited number of parts per component.

Procedure To add a part to the component;

1. Choose New Part from the Component menu.

See also

Remove Part

Remove Part

Summary Remove a part from the currently displayed component.

To Launch Alt C T

Purpose The Remove Part command is used to delete a part from a multiple part component.

Outcome The currently displayed part will be deleted from the component.

For example, using this command with the 74LS00 component which is in the TTL library, would delete one of its four gates.

If the component has only one part, then this command will remove the component from the library.

Comments You can use the Edit Undo command (shortcut: ALT+BACKSPACE) to restore the deleted part.

Procedure To remove a part from a multi part component;

1. Choose Remove Part from the Component menu.

See also

New Part

Next Part

Summary Show the next part of the currently displayed component.

To Launch Alt C O P

Purpose The Next Part command is used to show the next part of a multiple part component.

Outcome The next part of the component will be displayed. For example, using this command with the 74LS00 component which has four parts, would display its next gate.

Comments The system will beep if you try to go beyond the number of defined parts of a component. Use the Prev Part command to move back through the list of parts.

The Document Title bar displays the component name and the number of the currently displayed part, for example, 74LS00 1/4.

Procedure To view the next part in a multi part component;

1. Choose Component and then Next Part from the Component menu or click on the forward arrow > in the Part section of the Main Panel.

See also

Previous Part

Previous Part

Summary Show the previous part of the currently displayed component.

To Launch Alt C O V

Purpose The Previous Part command is used to show the previous part of a multiple part component.

Outcome The previous part of the component will be displayed. For example, using this command with the 74LS00 component which has four parts, would display its previous gate.

Comments The Document Title bar displays the component name and the number of the currently displayed part, for example, 74LS00 3/4.

Procedure To view the previous part in a multi part component;

1. Choose Component and then Prev Part from the Component menu or click on the back arrow < in the Part section of the Main Panel.

See also

Next Part

New Component

Summary Add a new component to library.

To Launch Alt C N - 

Purpose The New Component command is used to create a new component within the current library.

Outcome When the New Component command is used an empty component is created with the name "Component_1." If this name already exists then the name will increment to "Component_2."

Comments Use the Rename Component command from the Component menu to change the name of the created component. Component library sizes are virtually unlimited and are only restricted by available memory.

Procedure To add a new component to the library;

1. Choose New Component from the Component menu.

See also

Remove Component
Rename Component

Next Component

Summary Show the next component in the library.

To Launch Alt C O N

Purpose The Next Component command is used to show the next component in the library.

Comments The Document Title bar displays the component name and the number of the currently displayed part, for example, 74LS04 1/6.

Procedure To view the next component in the library;

1. Choose Component and then Next Component from the Component menu or click on the forward arrow > in the Components section of the Main Panel.

The next component in the library will be displayed.

See also

Previous Component

Remove Component Name

Summary Remove component name from group.

To Launch Alt C V

Purpose The Remove Component Name command is used to remove a name from a component group listing.

Outcome The selected name within the Group window of the Main Panel will be deleted from the list. If the component has only one name in its group then this command will remove the component from the library.

Comments You can use the Edit Undo command (shortcut: ALT+BACKSPACE) to restore the name to the component group listing.

Procedure To remove a name from the component group;

1. Choose Remove Component Name from the Component menu or click the Del button in the Group section of the Main Panel.

See also

Remove Component

Add Component Name

Summary Add component name to group.

To Launch Alt C A

Purpose The Add Component Name command is used to add a name to a component group listing.

Outcome The component name will be added to the component group and will be listed within the Group window of the Main Panel.

Comments Component group is a set of components which share the same graphical model in a library.

You can use the Remove Duplicates command from the Component menu to remove any duplicate components from the current library.

Procedure To add a name to the component group;

1. Choose Add Component Name from the Component menu or click the "Add" button in the Group section of the Main Panel.

2. Type the component name in the New Component Name dialog box and press ENTER or click OK.

See also

New Component

Previous Component

Summary Show the previous component in the library.

To Launch Alt C O P

Purpose The Previous Component command is used to show the previous component in the library.

Comments The Document Title bar displays the component name and the number of the currently displayed part, for example, 74LS04 1/6.

Procedure To view the previous component in the library;

1. Choose Component and then Prev Component from the Component menu or click on the back arrow < in the Components section of the Main Panel.

The previous component in the library will be displayed.

See also

Next Component

First Component

Summary Show the first component in the library.

To Launch Alt C O F

Purpose The First Component command is used to move the focus to the top of the component library listing and display the first component in the library.

Procedure To view the first component in the library;

1. Choose Component and then First Component from the Component menu or click on the back arrows << in the Components section of the Main Panel.

The selection bar within the Components window of the Main Panel will jump to the top of the library.

See also

Last Component

Last Component

Summary Show the last component in the library.

To Launch Alt C O L

Purpose The Last Component command is used to move the focus to the bottom of the component library listing and display the last component in the library.

Procedure To view the last component in the library;

1. Choose Component and then Last Component from the Component menu or click on the forward arrows >> in the Components section of the Main Panel.

The selection bar within the Components window of the Main Panel will jump to the bottom of the library.

See also

First Component

Show Normal

Summary Display component in standard ANSI notation.

To Launch Alt C H

Purpose The Show Normal command is used to switch the component display mode to standard ANSI schematic notation.

Comments Depending upon the part, components can be displayed using standard ANSI schematic notation, the De-Morgan equivalent (for gates) and IEEE standard notation.

You may also press one of the buttons in the Mode field on the Main Panel to change the display notation. Refer to the User Guide for detailed descriptions about Component and Part management.

Procedure To switch the component display mode to normal:

1. Choose Show Normal from the Component menu.

See also

Show De-Morgan
Show IEEE

Show De-Morgan

Summary Command Summary.

To Launch Alt C G

Purpose The Show De-Morgan command is used to switch the component display mode to De-Morgan notation.

Comments Depending upon the part, components can be displayed using standard ANSI schematic notation, the De-Morgan equivalent (for gates) and IEEE standard notation.

You may also press one of the buttons in the Mode field on the Main Panel to change the display notation. Refer to the User Guide for detailed descriptions about Component and Part management.

Procedure To switch the component display mode to De-Morgan:

1. Choose Show De-Morgan from the Component menu.

See also

Show Normal
Show IEEE

Show IEEE

Summary Command Summary.

To Launch Alt C I

Purpose The Show IEEE command is used to switch the component display mode to IEEE notation.

Comments Depending upon the part, these can be displayed using standard ANSI schematic notation, the De-Morgan equivalent (for gates) and IEEE standard notation.

You may also press one of the buttons in the Mode field on the Main Panel to change the display notation. Refer to the User Guide for detailed descriptions about Component and Part management.

Procedure To switch the component display mode to IEEE:

1. Choose Show IEEE from the Component menu.

See also

Show Normal
Show De-Morgan

Toggle Hidden Pins

Summary Show/Hide hidden pins in component.

To Launch Alt O S

Purpose The Toggle Hidden Pins command is used to show or hide all the pins that have the attribute of Hidden.

Comments The Toggle Hidden Pins command does not change the Hidden attribute of pins. The Hidden attribute can be changed via the Pin dialog box. Use the Change command or simply double click on the pin with the mouse.

Procedure To show/hide the hidden pins in the currently displayed component;

1. Choose Show Hidden Pins from the Options menu or click on the "Hidden Pins" check box in the Main Panel.

If this option is enabled then all the hidden pins on the displayed component will be shown.

See also

Place Part

Summary Adds a new part to the current component.

Purpose The Place Part command is used to place the currently selected part into an active schematic workspace.

Outcome After the Part is placed, your focus will remain in the Schematic Library Editor main window.

Comments You can also place parts from the Component Browser in the schematic editor by clicking the Place button. The Component Browser allows browsing of component names, placement of parts onto the current workspace, renaming of part designators and jumping to a part that has been placed on a workspace in a project.

Procedure To place a part into the current open schematic sheet:

Make sure the Advanced Schematic Editor application is running.

1. Select the part you wish to place by typing in the name of the part in the Mask field of the Main Panel. You can also use the "<","<<", ">>" and ">" buttons on the Main Panel to locate the part you want.

Make sure the name of the part you wish to place is highlighted.

2. Press the Place button on the Main Panel.

Your focus will shift to the Advanced Schematic Editor.

While moving the part, you can rotate it 90 degrees around the cursor by pressing the spacebar and flip it along its x or y axis by pressing the x and y keys. Press TAB to manually change the default values of the Part by typing new values directly into a dialog box.

4. Position the cursor where the Part is to be placed and press ENTER or LEFT MOUSE.

See also

Run Schematic Capture

Rename Component

Summary Change the name of the current component.

To Launch Alt C E

Purpose The Rename Component command is used to change the name of components in the current library.

Comments Schematic Library Editor allows you to have components in the library with the same name. However, when loading a file into the Schematic Editor the first component in the library will be used.

You can use the Remove Duplicates command from the Component menu to remove any duplicate components from the current library.

Procedure To rename the current component;

1. Choose Rename Component from the Component menu.

2. Type a new component name in the New Component Name dialog box and press ENTER or click OK.

The new component name will be listed within the Components window of the Main Panel.

Remove Component

Summary Remove component from the current library.

To Launch Alt C R

Purpose The Remove Component command is used to remove components from the current library.

Comments You can use the Edit Undo command (shortcut: ALT + BACKSPACE) to restore the deleted component.

Procedure To remove a component from the library;

1. Choose Remove Component from the Component menu.

The currently displayed component will be removed from the library.

See also

Remove Duplicates

Component Description

Summary Change component description fields.

To Launch Alt C P

Purpose The Component Description command is used to attach a description, component footprint names and other information to a library component.

Outcome The information entered in the Component Text Fields dialog box will be saved with the other library information about the current component when the library is saved in Protel format.

Comments The information entered for the Text Field descriptions with a library component and cannot be altered in Schematic Library Editor. However, the information can be accessed when generating a Bill of Materials in Advanced Schematic Editor.

Description This field is intended to provide a text description of the component's function for reference purposes. Supports up to 255 characters, including spaces.

Default Designator Defines the default designator for placement of the component in a schematic sheet. Letters numbers or a "?" may be used in this field. Supports up to 255 characters, including spaces.

Foot Prints These four fields may be used to indicate the names of component footprints to be used in the Protel for Windows PCB Design System. Supports up to 255 characters, including spaces)

Text Fields These eight fields may be used for notation of any purpose. Supports up to 255 characters, including spaces. They are in additional to the text fields that are editable within Schematic Library Editor and specific to each component.

Procedure To edit the description text for a component:

Make sure the component you wish to attach descriptions to is highlighted in the component list box on the Main Panel.

1. Choose Description from the Component menu.

The Component Text Fields dialog box will appear.

2. Type information into the desired fields

3. Press OK to return to the editor.

See also

Remove Duplicates

Summary Remove duplicate components from library.

To Launch Alt C D

Purpose The Remove Duplicates command is used to remove/delete duplicate components from the current library. The first component name in the library is saved and all other components with the same name are removed.

Comments This command is useful especially when using the file merge command, where all components from one library are copied into another library.

Procedure To remove duplicate components from the current library:

1. Choose Remove Duplicates from the Component menu.

Any duplicated components in the library will be removed.

See also

File Merge
Remove Component

Run Control Panel

Summary Run the Windows Control Panel Program.

To Launch T W P

Purpose This command is used to launch Windows Control Panel utility directly from Advanced Schematic. Tools menu commands provide convenient ways to launch frequently used applications and utilities.

Outcome The Windows Control Panel utility will be launched.

Comments This is not a user-selectable Run command option.

Procedure Choose the Windows Tools command from the Tools menu then choose Control Panel.

See also

Setup User Programs

Run Windows Setup

Summary Run the Windows Setup Program.

To Launch T W S

Purpose This command is used to launch the Windows Setup utility directly from Advanced Schematic. Tools menu commands provide convenient ways to launch frequently used applications and utilities.

Outcome The Windows Setup utility will be launched.

Comments This is not a user-selectable Run command option.

Procedure Choose the Windows Tools command from the Tools menu then choose Windows Setup.

See also

Setup User Programs

Run Text Editor

Summary Run the user specified text editor. (Defaults To Notepad.Exe).

To Launch T W T

Purpose This command is used to launch a user-specified text editor directly from Advanced Schematic. Run menu options provide a convenient way to launch frequently used applications and utilities.

Outcome Launches the specified text editor application.

Comments User can specify the Text Editor application under the Tools-Setup command.

Procedure Choose the Windows Tools command from the Tools menu then choose Text Editor.

See also

Setup User Programs

Run Picture Editor

Summary Run the user specified Paint/Draw program (Default is Windows Paint).

To Launch T W I

Purpose This command is used to launch a user-specified paint/draw application directly from Advanced Schematic. Tools menu commands provide convenient ways to launch frequently used applications and utilities.

Outcome Launches the specified Picture Editor application.

Comments User can specify the Picture Editor application under the Tools-Setup command.

Procedure Choose the Windows Tools command from the Tools menu then choose Picture Editor.

See also

Setup User Programs

Run File Manager

Summary Run the Windows File Manager Program.

To Launch T W F

Purpose This command is used to launch the Windows File Manager utility directly from Advanced Schematic. Tools menu commands provide convenient ways to launch frequently used applications and utilities.

Outcome Launches the Windows File Manager utility.

Comments This is not a user-selectable Run command option.

Procedure Choose the Windows Tools command from the Tools menu then choose File Manager.

See also

Setup User Programs

Run Calculator

Summary Run the Windows Calculator Program.

To Launch T W C

Purpose This command is used to launch the Windows Calculator utility directly from Advanced Schematic. Tools menu commands provide convenient ways to launch frequently used applications and utilities.

Outcome Launches the Windows Calculator utility.

Comments This is not a user-selectable Run command option.

Procedure Choose the Windows Tools command from the Tools menu then choose Calculator.

See also

Setup User Programs

Run Clock

Summary Run the Windows Clock Program.

To Launch T W L

Purpose This command is used to launch the Windows Clock utility directly from Advanced Schematic. Tools menu commands provide convenient ways to launch frequently used applications and utilities.

Outcome Launches the Windows Clock utility.

Comments This is not a user-selectable Run command option.

Procedure Choose the Windows Tools command from the Tools menu then choose Clock.

See also

Setup User Programs

Run User Program 1

Summary Run user specified Program 1.

To Launch T U 1

Purpose This command is used to launch an application directly from Advanced Schematic. Tools menu commands provide convenient ways to launch frequently used applications and utilities.

Outcome Launches the specified User Program (application/utility) #1.

Comments User can specify the User Program #1 application under the Tools-Setup command.

Procedure Choose the User Tools command from the Tools menu then choose User Program 1.

See also

Setup User Programs

Run User Program 2

Summary Run user specified Program 2.

To Launch T U 2

Purpose This command is used to launch an application directly from Advanced Schematic. Tools menu commands provide convenient ways to launch frequently used applications and utilities.

Outcome Launches the specified User Program (application) #2.

Comments User can specify the User Program #2 application under the Tools-Setup command.

Procedure Choose the User Tools command from the Tools menu then choose User Program 2.

See also

Setup User Programs

Run User Program 3

Summary Run user specified Program 3.

To Launch T U 3

Purpose This command is used to launch an application directly from Advanced Schematic. Tools menu commands provide convenient ways to launch frequently used applications and utilities.

Outcome Launches the specified User Program (application) #3.

Comments User can specify the User Program #3 application under the Tools-Setup command.

Procedure Choose the User Tools command from the Tools menu then choose User Program 3.

See also

Setup User Programs

Run User Program 4

Summary Run user specified Program 4.

To Launch T U 4

Purpose This command is used to launch an application directly from Advanced Schematic. Tools menu commands provide convenient ways to launch frequently used applications and utilities.

Outcome Launches the specified User Program (application) #4.

Comments User can specify the User Program #4 application under the Tools-Setup command.

Procedure Choose the User Tools command from the Tools menu then choose User Program 4.

See also

Setup User Programs

Run CSV Editor

Summary Run the user specified program for editing CSV files.

To Launch T W V

Purpose This command is used to launch an application directly from Advanced Schematic. Run menu options provide a convenient way to launch frequently used applications and utilities.

Outcome Launches the specified CSV Editor application.

Comments User can specify the CSV Editor application under the Tools-Setup command.

Some DOS applications cannot be run from inside Windows. Attempting to launch incompatible DOS applications from the Tools menu could cause unexpected results, including system hangs.

Procedure Choose the Windows Tools command from the Tools menu then choose CSV Editor.is chosen.

See also

Setup User Programs

Setup User Programs

Summary Setup user specific Programs to use with Tools Menu.

To Launch T S

Purpose This command is used to set-up applications to be launched directly from Advanced Schematic Library Editor. Tools menu options provide a convenient way to launch frequently used applications and utilities.

Outcome Opens the Setup Run Options dialog box.

Comments Some DOS applications cannot be run from inside Windows. Attempting to launch incompatible DOS applications from the Tools menu could cause unexpected results, including system hangs.

Procedure 1. Choose the Setup command from the Tools menu to open the Setup Run Options dialog box.

2. Type the path (if necessary) and application name for each option in the appropriate box. Options include:

Schematic Capture - Editor for schematic capture files. Default is SCH.EXE.

Text editor - Editor for Reports (other than BOM CSV format). Default is NOTEPAD.EXE.

Picture editor - Editor for Image files. Default is PBRUSH.EXE.

CSV editor - Editor for BOM (CSV portion) reports. Default is NOTEPAD.EXE.

User Program 1 to 4 - any user specified application or utility. None specified by default.

See also

Run File Manager
Run Control Panel
Run Calculator
Run Clock
Run Notepad
Run Text Editor
Run Picture Editor
Run CSV Editor
Run User Program 1
Run User Program 2
Run User Program 3
Run User Program 4

Run Notepad

Summary Run the Windows Notepad program.

To Launch T W N

Purpose This command is used to launch the Windows Notepad program directly from Advanced Schematic. Tools menu commands provide convenient ways to launch frequently used applications and utilities.

Outcome Launches the Windows Notepad utility.

Comments This is not a user-selectable Run command option.

Procedure Choose the Windows Tools command from the Tools menu then choose Notepad.

See also

Setup User Programs

Popup Component Menu

Summary Popup the Component menu.

Purpose The Popup Component Menu command is used to directly access the Component menu at the current cursor position.

Procedure To popup the Component menu at the current cursor position.

1. By default, press C.

You can change the default hot key by using the Options Hot Keys command.

See also

Hot Keys

Component Rule Check

Summary Validate changes to current library.

To Launch Alt F R R

Purpose Screens the current active library for user-specified errors.

Outcome Generates a report file listing library (or component) errors for the current active library.

Comments When checking for duplicate pins, only the first instance will be reported for each component. If more than 1 duplicate exisits, this will not be reported separately.

Procedure When you choose this command, each component description in the current library is scanned. Errors and warnings are generated and displayed in a text file which is opened using Notepad or your designated text editing application (see Run command for details).

Library errors include:

Duplicate Component Names

Duplicate Pins on a component

Missing Component Description

Missing Default Designators

Missing Footprints (if all fields are empty)

Missing Pin Names on a component

Missing Pin Numbers on a component

Popup Move Menu

Summary Popup the Edit Move menu.

Purpose The Popup Move Menu command is used to directly access the Move menu at the current cursor position.

Procedure To popup the Move menu at the current cursor position.

1. Press M. (Default hot key assignment)

You can change the default hot key by using the Options Hot Keys command.

See also

Hot Keys

Toggle Command Status Bar

Summary Turn the Command Status Line on or off.

Purpose The Toggle Command Status Bar command is used to turn the lower half of the Status line (which displays the current command status) on or off at the bottom of the Advanced Schematic application window. Turning off the process status line shows more screen region.

Outcome Choosing Command Status Bar from the Options menu will turn the Command Status line on or off. The Status line must be enabled for this command to be effective.

Comments The Command status line displays the name of the currently running command.

Procedure To turn the Command Status line on or off:

1. Choose Command Status Bar from the Options menu.

Find and Replace Text

Summary Search for and replace text strings.

To Launch Ctrl G - Alt E A

Purpose The Search for and replace text command allows you to conveniently locate and replace specific text that has been placed on any worksheet without having to zoom, pan or scroll through multiple screens.

Outcome Matching text will be replaced by the new replacement text.

Comments The wildcard character "*" can be used to extend the definition of

target strings. For example, S* will limit the fields to strings

beginning with S, etc. Wildcards are case in-sensitive.

Braces "" and "" can also be used used to define the rules for

text replacement. Defining the change follows this syntax: oldtext=newtext. This means change portion of the string "oldtext" to "newtext". You can use multiple sets of brackets to define complex replacements. In this case the leftmost replacement is made, then the next on, etc. Although this is very powerful, you must take care, because the first change can effect subsequent replacements, possibly generating an unexpected result. Any mistakes can be corrected with the Undo command, however.

The find and replace text command can find and replace text

anywhere in a component or across a whole library. Extends across all objects with visible text, including Pins.

Procedure To search and replace text:

1. Choose Replace Text from the Edit menu. The Text Find And Replace dialog box opens.

2. Type the Text To Find string.

3. In the New Text field, type the replacement text.

4. Choose the Scope for the change. Sheet Scope - Current Document or All documents. Selection - Selected Objects, Deselected Objects, All Objects.

5. Choose any other Options. Case Sensitive, Prompt On Replace.

6. Click OK to perform the text replacement.

See also

Find Next Text

Summary Search for and jump to the next matching text string.

To Launch F3 - Alt E X

Purpose The Find Next Text command allows you to conveniently locate specific text that has been placed on any worksheet without having to zoom, pan or scroll through multiple screens

Comments You can also Find and Replace text.

Procedure To find the next occurence of a string:

1. Choose Find Next from the Edit menu.

See also

Find Text

Summary Search for and jump to a text string on the workspace..

To Launch Ctrl F - Alt E F

Purpose The Find Text command allows you to conveniently locate specific text that has been placed on any workspace without having to zoom, pan or scroll through multiple screens.

Comments The wildcard character "*" can be used to extend the definition of

target strings. For example, S* will limit the fields to strings

beginning with S, etc. Wildcards are case in-sensitive.

The find text command can find text anywhere in a component or across a library. Extends across all objects with visible text, including Pins.

Procedure To search for text:

1. Choose the Find Text from the Edit menu.

2. Type the search text into the Text to find field in the Find Text Dialog box.

3. Choose the Scope for the change.

Changes can be applied to the Current Docuement Only or to All

Open Documents. Objects with text to be changed can be

restricted to selected or unselected items.

4. Choose any other Options.

Jump New Location

Summary Type in and jump to a new location on the worksheet.

To Launch Alt E J L

Purpose The Jump Location command is used to move the cursor to a specified location by typing in X and Y coordinates. This can save you from having to constantly zoom in and out to navigate around your component design and is particularly useful for large or complex components.

Outcome The cursor is placed on the specified location.

Procedure To move the cursor to a specified location.

1. Choose Jump Location from the Edit menu.

2. Type an X coordinate (a distance from the horizontal center of the worksheet), the default is the current X position.

3. Type a Y coordinate (a distance from the vertical center of the worksheet), the default is the current Y position.

3. Click OK to jump to the specified location.

See also

Jump To Origin

Popup Jump Menu

Summary Popup the Edit Jump menu.

Purpose The Popup Jump Menu command is used to directly access the Edit Jump menu at the current cursor position.

Comments You can change the default hot key by using the Options Hot Keys command.

Procedure To popup the Edit Jump menu at the current cursor position.

1. Press the J keyboard hot key. (Default hot key assignment.)

See also

Hot Keys

Change Object Graphically or Move

Summary Select and change objects on a library component.

Purpose This command is used to set the focus on an object. If the object already has the focus then this command interactively changes the object by physically moving the position of the objects handles.

Comments The Change Single Object Graphically or Move command can only be accessed using the mouse. This can be changed using setup hotkeys command. You can use the Edit Undo command (shortcut: ALT+BACKSPACE) to restore the edited objects.

Procedure To move an object;

1. Move the cursor over the object to be graphically edited and click and hold LEFT MOUSE.

If the cursor is over a handle then the handle can be move, otherwise the whole object will be moved.

2. Use the mouse to move the object.

See also

Change

Change Object Graphically or Set Focus

Summary Select and change objects on a library component.

Purpose This command is used to set the focus on an object. If the object already has the focus then this command interactively changes the object by physically moving the position of the objects handles.

Comments The Change Single Object Graphically command can only be accessed using the mouse. This can be changed using setup hotkeys command. You can use the Edit Undo command (shortcut: ALT+BACKSPACE) to restore the edited objects.

Procedure To set the focus on an object;

1. Move the cursor over the object to be graphically edited and click LEFT MOUSE. The object will then be displayed with graphical editing handles.

2. Use the mouse to graphically change the object.

See also

Change

Align Objects

Summary Align selected objects using alignment dialog box.

To Launch Alt E G A

Purpose The Align Objects command is used to align a group of selected objects on both x and y axis at the same time. Objects can be aligned by their left/right/top/bottom sides, center alignment, distributed horizontally or vertically or moved to the placement grid. You can align objects on one axis, by choosing the other Align menu commands.

Outcome The selection will be aligned according to the chosen options.

Procedure To align selected objects using the Align Objects dialog box:

1. Add all items to be aligned to the current selection, making sure that only items to be aligned are selected. You can use the Edit-DeSelect-All command (shortcut: x, a) to clear the current selection.

2. Choose Align from the Edit menu and then choose the Align command.

3. Select the desired horizontal and/or vertical alignment combination. You can also choose the Align To Grid option to constrain alignment to the nearest grid point.

4. Click OK to close the dialog box and align the selection.

See also

Distribute equally along vertical axis
Distribute equally along horizontal axis
Center objects around vertical axis
Center objects around horizontal axis
Align objects on bottom
Align objects on top
Align objects on right
Align objects on left

Align objects on left

Summary Align selected objects on the left side of their bounding rectangle.

To Launch Ctrl L - Alt E G L

Purpose The Align objects on left command is used to align a group of selected objects along the left side of their bounding rectangle.

Outcome All selected objects will be aligned on the left edge of their bounding rectangle.

Procedure To align objects using the Align Left command:

1. Add all items to be aligned to the current selection, making sure that only items to be aligned are selected. You can use the Edit-DeSelect-All command (shortcut: x, a) to clear the current selection.

2. Choose Align from the Edit menu and then choose the Align Left command.

See also

Distribute equally along vertical axis
Distribute equally along horizontal axis
Center objects around vertical axis
Center objects around horizontal axis
Align objects on bottom
Align objects on top
Align objects on right

Align objects on right

Summary Align selected objects on the right side of their bounding rectangle.

To Launch Ctrl R - Alt E G R

Purpose The Align objects on right command is used to align a group of selected objects along the right side of their bounding rectangle.

Outcome All selected objects will be aligned on the right edge of their bounding rectangle.

Procedure To align objects using the Align Right command:

1. Add all items to be aligned to the current selection, making sure that only items to be aligned are selected. You can use the Edit-DeSelect-All command (shortcut: x, a) to clear the current selection.

2. Choose Align from the Edit menu and then choose the Align Right command.

See also

Distribute equally along vertical axis
Distribute equally along horizontal axis
Center objects around vertical axis
Center objects around horizontal axis
Align objects on bottom
Align objects on top
Align objects on left

Center objects around horizontal axis

Summary Center objects around the vertical center line of the bounding rectangle.

To Launch Ctrl H - Alt E G C

Purpose The Center objects around horizontal axis command is used to align a group of selected objects along their horizontal center l

Outcome All selected objects will be aligned along their horizontal center line.

Procedure To align objects using the Center Horizontal command:

1. Add all items to be aligned to the current selection, making

sure that only items to be aligned are selected. You can use

the Edit-DeSelect-All command (shortcut: x, a) to clear the

current selection.

2. Choose Align from the Edit menu and then choose the Center Horizontal command.

See also

Distribute equally along vertical axis
Distribute equally along horizontal axis
Center objects around vertical axis
Align objects on bottom
Align objects on top
Align objects on right
Align objects on left

Distribute equally along horizontal axis

Summary Distribute equally the selected objects along the horizontal axis.

To Launch Ctrl Shift H - Alt E G D

Purpose This command is used to space a group of selected objects equally along the horizontal axis.

Outcome All selected objects will be distributed equally along the horizontal axis.

Procedure To distribute objects equally along the horizontal axis:

1. Add all items to be distributed to the current selection, making

sure that only items to be distributed horizontally are selected. You can use the Edit-DeSelect-All command (shortcut: x, a) to clear the

current selection.

2. Choose Align from the Edit menu and then choose the Distribute Horizontally command.

See also

Distribute equally along vertical axis
Center objects around vertical axis
Center objects around horizontal axis
Align objects on bottom
Align objects on top
Align objects on right
Align objects on left

Align objects on top

Summary Align selected objects on the top side of their bounding rectangle.

To Launch Ctrl T - Alt E G T

Purpose The Align objects on top command is used to align a group of selected objects along the top edge of their bounding rectangle.

Outcome All selected objects will be aligned on the top edge of their bounding rectangle.

Procedure To align objects using the Align Top command:

1. Add all items to be aligned to the current selection, making sure that only items to be aligned are selected. You can use the Edit-DeSelect-All command (shortcut: x, a) to clear the current selection.

2. Choose Align from the Edit menu and then choose the Align Left command.

See also

Distribute equally along vertical axis
Distribute equally along horizontal axis
Center objects around vertical axis
Center objects around horizontal axis
Align objects on bottom
Align objects on right
Align objects on left

Align objects on bottom

Summary Align selected objects on the bottom side of their bounding rectangle.

To Launch Ctrl B - Alt E G B

Purpose The Align objects on bottom command is used to align a group of selected objects along the bottom edge of their bounding rectangle.

Outcome All selected objects will be aligned on the bottom edge of their bounding rectangle.

Procedure To align objects using the Align Bottom command:

1. Add all items to be aligned to the current selection, making sure that only items to be aligned are selected. You can use the Edit-DeSelect-All command (shortcut: x, a) to clear the current selection.

2. Choose Align from the Edit menu and then choose the Align Bottom command.

See also

Distribute equally along vertical axis
Distribute equally along horizontal axis
Center objects around vertical axis
Center objects around horizontal axis
Align objects on top
Align objects on right
Align objects on left

Center objects around vertical axis

Summary Center objects around the vertical center line of the bounding rectangle.

To Launch Ctrl V - Alt E G V

Purpose The Center objects around vertical axis command is used to align a group of selected objects along their vertical center line.

Outcome All selected objects will be aligned along their vertical center line.

Procedure To align objects using the Center Vertical command:

1. Add all items to be aligned to the current selection, making sure that only items to be aligned are selected. You can use the Edit-DeSelect-All command (shortcut: x, a) to clear the current selection.

2. Choose Align from the Edit menu and then choose the Center Vertical command.

See also

Distribute equally along vertical axis
Distribute equally along horizontal axis
Center objects around horizontal axis
Align objects on bottom
Align objects on top
Align objects on right
Align objects on left

Distribute equally along vertical axis

Summary Distribute equally the selected objects along the vertical axis.

To Launch Ctrl Shift V - Alt E G I

Purpose This command is used to space a group of selected objects equally along the vertical axis.

Outcome All selected objects will be distributed equally along the vertical axis.

Procedure To distribute objects equally along the vertical axis:

1. Add all items to be distributed to the current selection, making

sure that only items to be distributed vertically are selected. You can use the Edit-DeSelect-All command (shortcut: x, a) to clear the

current selection.

2. Choose Align from the Edit menu and then choose the Distribute Verticallly command.

See also

Distribute equally along horizontal axis
Center objects around vertical axis
Center objects around horizontal axis
Align objects on bottom
Align objects on top
Align objects on right
Align objects on left

Open Environment Configuration File

Summary Open and load an environment configuration file.

To Launch O O

Purpose This command is used to open and load an environment configuration file. This includes all options, preferences and the current set of open library files.

Outcome Once an environment configuration file has been selected, the current environment settings will be cleared and the new settings will be loaded, this includes all options, preferences and any files that were open when the original environment file was saved.

Comments All current settings will be lost and all files will be closed. Advanced Schematic Library Editor will prompt you to save any open files first.

Procedure To open a previously saved environment configuration file:

1. Choose Open from the Options menu.

2. Type the filename (include the full path, if different from the path listed after Directory)

3. Click OK to open the file.

You can also double-click on the desired filename in the Files window, if any. To change directories, click under any of the options listed in the Directories window.

See also

Save Environment Configuration File

Save Environment Configuration File

Summary Save current environment configuration to a file.

To Launch O A

Purpose The File Save As command can be used to save the current environment configuration to a file. This includes all options, preferences and the list of all currently open files.

Outcome When you select the Options Save As command, the Save Environment File As dialog box will open. Once you click OK the file will be saved to the new file name and/or directory that you specified.

Procedure To save the current environment configuration to a new path, file name.

1. Choose Save As from the Options menu.

2. Type a new file name for the environment configuration.

3. Click OK to save the file.

To change directories, click under any of the options listed in the Directories window.

See also

Open Environment Configuration File

Setup Autopan

Summary Setup Autopan options.

To Launch Alt O U

Purpose This command is used to configure the effect of Autopanning aroung the screen.

Outcome Opens the Setup Autopan dialog box, and allows customization of autopanning around the screen.

Comments When the Options Auto Pan feature is enabled, your

view of the current window will automatically pan up-down-left-

right as you place, move or draw objects in the workspace. Panning

takes place when the cursor reaches the edge of the display when

placing, moving or re-sizing objects.

Procedure To change the autopan settings:

1. Choose Auto-Pan from the Options menu

2. The Setup Autopan dialog box will open.

Three options can be set within this dialog box;

Style - while autopanning the cursor can jump by a pre-defined step size, a full screen at a time with the cursor re-centering in the middle of the screen or can be disabled.

Step Size - distance the cursor will jump while autopanning, the autopan Style must be set to Fixed Size Jump.

Shift Step Size - distance the cursor will jump while autopanning and holding down the SHIFT key, the autopan Style must be set to Fixed Size Jump.

Popup Alignment Menu

Summary Popup the Alignment menu.

Purpose The Popup Alignment Menu command is used to directly access the Align menu at the current cursor position.

Comments You can change the default hot key by using the Options Hot Keys command.

Procedure To popup the Alignment menu at the current cursor position.

1. Press the A keyboard hot key. (Default hot key assignment)

See also

Hot Keys

Memory Monitor Setup

Summary Setup low memory/resources warning thresholds..

To Launch O Y

Purpose This command is used to monitor the system memory and system resources.

In the Memory Monitor dialog box the user can define minimum levels for system memory and system resources. These thresholds are used by the application to determine when a warning message is to be issued.

Outcome The following are options that can be set within the Memory Monitor dialog box.

Frequency - specifies when memory is checked. By default it is checked after every 16KB of memory is allocated.

Memory Warning - enables/disables the warning and specifies the memory threshold. By default the warning threshold is 2000KB.

System Resources Warning - enables/disables the warning and specifies the system resource threshold. By default the warning threshold is 30%.

Comments When either of these two options are enabled and the available memory or system resources fall below the defined thresholds a warning dialog box will open stating the problem and appropriate actions.

Note: When the system resources fall below 10% Windows can behave unexpectedly, therefore it is not recommended to set the resource threshold warning lower than 10%.

Procedure To define the warning thresholds for system memory and system resources.

1. Choose Memory Monitor from the Options menu.

2. Choose or type in new memory monitor settings.

3. Click OK to accept the new settings.

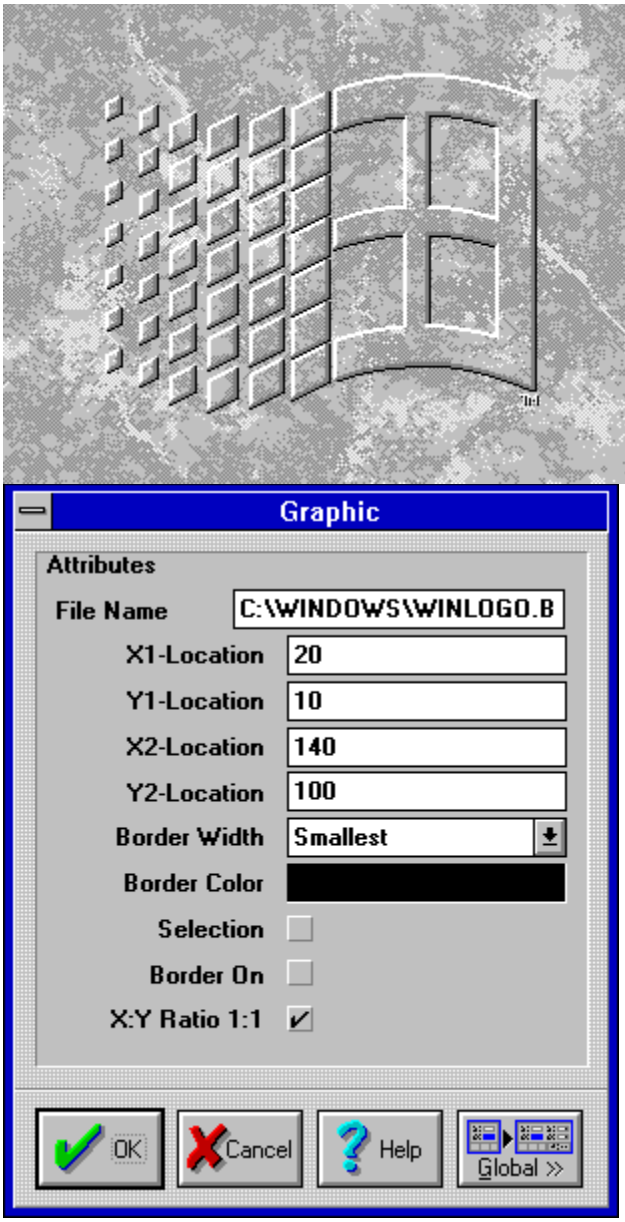
See also

Popup Info Menu

Data Primitives

Graphic

Overview Graphic images are special graphical objects, used for adding reference information to a library component, such as mechanical and electrical symbols or customized component images.



Fields

File Name The path and file name of the image file.

X1-Location The left hand edge reference point coordinate of the imported graphic image, along the x-axis.

Y1-Location The bottom edge reference point coordinate of the imported graphic image, along the y-axis.

X2-Location The right hand edge reference point coordinate of the imported graphic image, along the x-axis.

Y2-Location The top edge reference point coordinate of the imported graphic image, along the y-axis.

Border Width Specifies the border outline thickness of imported graphic images. There are four selectable border widths; smallest, small, medium and large.

Border Color The surrounding border of imported graphic images can be assigned a color. To assign a new color to the border, click in the color box to open the Color Selector dialog box. Schematic Library Editor will display all the available colors that your graphics card and monitor supports.

Selection Toggle the selection state of imported graphic images by turning this option on or off. If this option is on, the graphic image will be outlined in the selection color defined in the Options Preferences dialog box.

Selections are generally made three ways; using the option in a dialog box, using SHIFT+LEFT MOUSE to add (or remove) individual objects to the current selection and by using the Edit Select and De-select commands to define a selection group.

Border On The surrounding border of imported graphic images can be toggled on or off. If this option is enabled, the border around the graphical image will be displayed in the border color. If this option is off, then only the graphic image will be shown.

X:Y Ratio 1:1 Graphical images may be re-sized by moving one of its handles, when this option is on, the image retains the original x and y aspect ratio.

Comments Schematic Library Editor supports the import of the following image formats;

BMP - All uncompressed Bit Map images. Windows device-independent Bit Map format, introduced with Windows 3.0 and increasingly supported by Windows applications.

PCX - Paintbrush format, used in Windows Paintbrush and other paint programs and supported by many desktop publishing and graphics programs. Supported colors include, monochrome, 16 color, 256 color, 24-bit color.

TIFF - Tag Image File Format, supported by many desktop publishing programs. Supported compression types, uncompressed, LZW, Packbits, Modified Huffman encoding, CCITT Group 3 1D, CCITT Group 3 2D, CCITT Group 4. Supported colors include, monochrome, 16 color, 256 color, 24-bit color

GIF - All non-interlaced Graphic Image files.

EPS - Encapsulated postscript files with and without display images. If the EPS file does not contain a TIFF or Windows Metafile display image then the file name of the EPS image will be displayed.

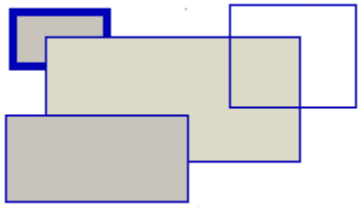
WMF - Only Windows Metafiles which conform to the Aldus Placeable Metafile Format are supported. Most applications which export or import Metafiles support this format.

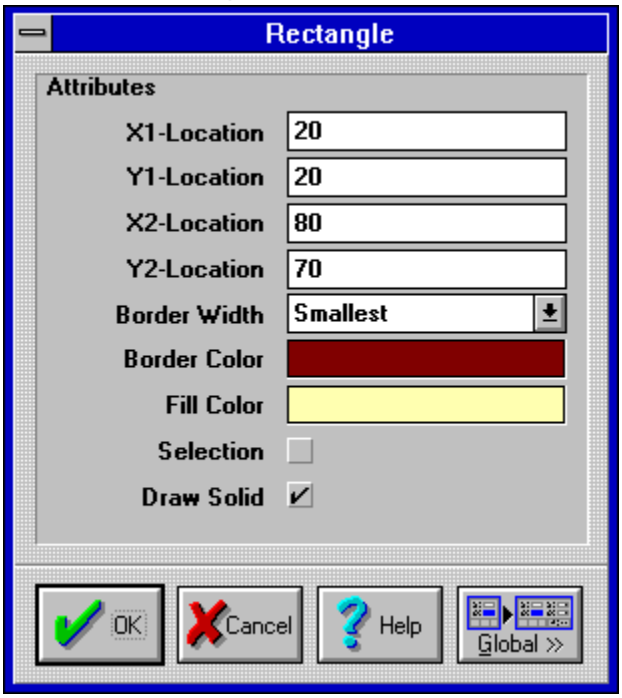
See also

Place Graphic

Rectangle

Overview Rectangles are filled or unfilled graphic elements, used for building library components, such as defining the component body or adding graphical symbols.





Fields

X1-Location The left hand edge reference point coordinate of the rectangle, along the x-axis.

Y1-Location The bottom edge reference point coordinate of the rectangle, along the y-axis.

X2-Location The right hand edge reference point coordinate of the rectangle, along the x-axis.

Y2-Location The top edge reference point coordinate of the rectangle, along the y-axis.

Border Width Specifies the border outline thickness of rectangles. There are four selectable border widths; smallest, small, medium and large.

Border Color The surrounding border of Rectangles can be assigned a color. To assign a new color to the border, click in the color box to open the Color Selector dialog box. Schematic Library Editor will display all the available colors that your graphics card and monitor supports.

Fill Color The inside area of Rectangles can be assigned a color. To assign a new color to the inside area, click in the color box to open the Color Selector dialog box. Schematic Library Editor will display all the available colors that your graphics card and monitor supports.

Selection Toggle the selection state of Rectangles by turning this option on or off. If this option is on, the Rectangle will be outlined in the selection color defined in the Options Preferences dialog box.

Selections are generally made three ways; using the option in a dialog box, using SHIFT+LEFT MOUSE to add (or remove) individual objects to the current selection and by using the Edit Select and De-select commands to define a selection group.

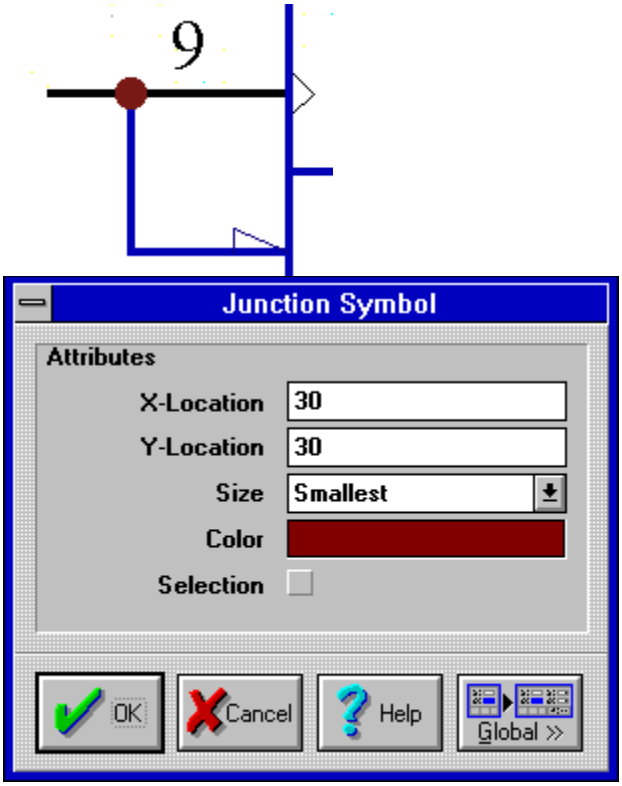
Draw Solid Schematic Library Editor allows you to turn the inside area color of Rectangles on or off. If this option is on, the inside area of the Rectangle will be displayed in the fill color. If this option is off, then only the Rectangles border will be shown.

See also

Place Rectangle
Round Rectangle

Junction Symbol

Overview Junction symbols are small circular objects used for building library components on the library workspace. Junction symbols are graphical only they have no electrical significance.



Fields

X-Location The reference point coordinate of the junction symbol, along the x-axis.

Y-Location The reference point coordinate of the junction symbol, along the y-axis.

Size Specifies the size of junction symbols. There are four selectable junction sizes; smallest, small, medium and large.

Color Junction symbols can be assigned a color. To assign a new color to the Junction, click in the color box to open the Color Selector dialog box. Schematic Library Editor will display all the available colors that your graphics card and monitor supports.

Selection Toggle the selection state of Junction symbols by turning this option on or off. If this option is on, the Junction will be outlined in the selection color defined in the Options Preferences dialog box.

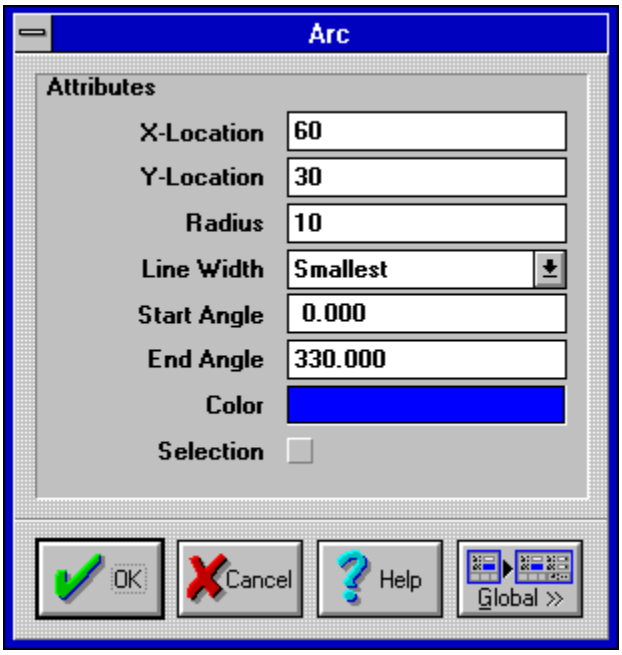
Selections are generally made three ways; using the option in a dialog box, using SHIFT+LEFT MOUSE to add (or remove) individual objects to the current selection and by using the Edit Select and De-select commands to define a selection group.

See also

Place Junction Symbol

Arc

Overview Arcs are circular line segments, used for building library components, such as defining component outlines or adding graphical symbols.



Fields

X-Location The center point coordinate of an arc, along the x-axis.

Y-Location The center point coordinate of an arc, along the y-axis.

Radius Schematic Library Editor allows you to change the radius of Arcs. The radius can be any integer (whole number) value. A value of 1 = 10mil = .254mm.

Line Width Specifies the line thickness of Arc segments. There are four selectable Arc widths; smallest, small, medium and large.

Start Angle Schematic Library Editor allows you to change the start angle of an Arc. The start angle can be any degree from 1 to 360, (Real number).

End Angle Schematic Library Editor allows you to change the end angle of an Arc. The start angle can be any degree from 1 to 360, (Real number).

Color Arcs can be assigned a color. To assign a new color to the Arc, click in the color box to open the Color Selector dialog box. Schematic Library Editor will display all the available colors that your graphics card and monitor supports.

Selection Toggle the selection state of Arcs by turning this option on or off. If this option is on, the

Arcs will be outlined in the selection color defined in the Options Preferences dialog box.

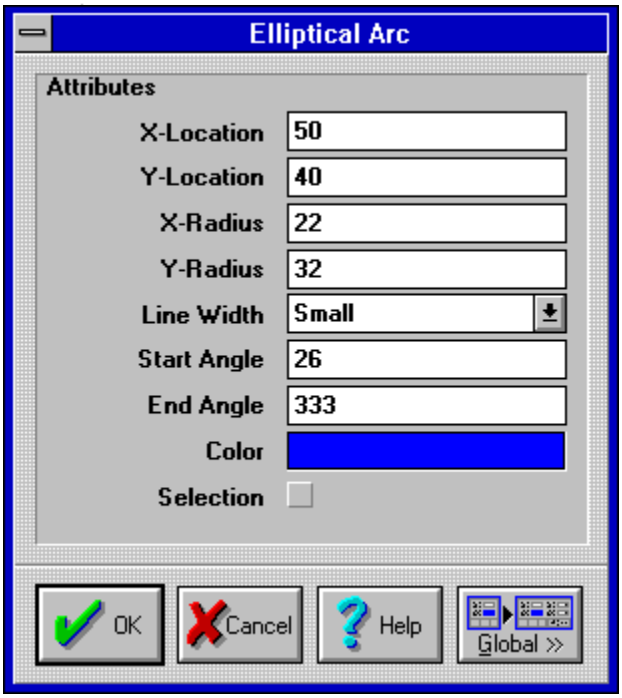
Selections are generally made three ways; using the option in a dialog box, using SHIFT+LEFT MOUSE to add (or remove) individual objects to the current selection and by using the Edit Select and De-select commands to define a selection group.

See also

Place Arc
Elliptical Arc

Elliptical Arc

Overview Elliptical Arcs are graphical objects with a user-defined line width and start/end coordinates. Elliptical Arcs are used for building library components, such as defining component outlines or adding graphical symbols.



Fields

X-Location The reference point coordinate of the elliptical arc, along the x-axis.

Y-Location The reference point coordinate of the elliptical arc, along the y-axis.

X-Radius Schematic Library Editor allows you to change the horizontal width of Elliptical Arcs. The x-radius can be any integer (whole number) value. A value of 1 = 10mil = .254mm.

Y-Radius Schematic Library Editor allows you to change the vertical height of Elliptical Arcs. The y-radius can be any integer (whole number) value. A value of 1 = 10mil = .254mm.

Line Width Specifies the line thickness of elliptical arc segments. There are four selectable elliptical arc widths; smallest, small, medium and large.

Start Angle Schematic Library Editor allows you to change the start angle of Elliptical Arcs. The start angle can be any degree from 1 to 360, (Real number).

End Angle Schematic Library Editor allows you to change the end angle of Elliptical Arcs. The start angle can be any degree from 1 to 360, (Real number).

Color Elliptical Arcs can be assigned a color. To assign a new color to the Elliptical Arc, click in

the color box to open the Color Selector dialog box. Schematic Library Editor will display all the available colors that your graphics card and monitor supports.

Selection Toggle the selection state of Elliptical Arcs by turning this option on or off. If this option is on, the Elliptical Arc will be outlined in the selection color defined in the Options Preferences dialog box.

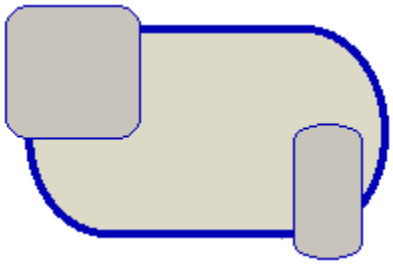
Selections are generally made three ways; using the option in a dialog box, using SHIFT+LEFT MOUSE to add (or remove) individual objects to the current selection and by using the Edit Select and De-select commands to define a selection group.

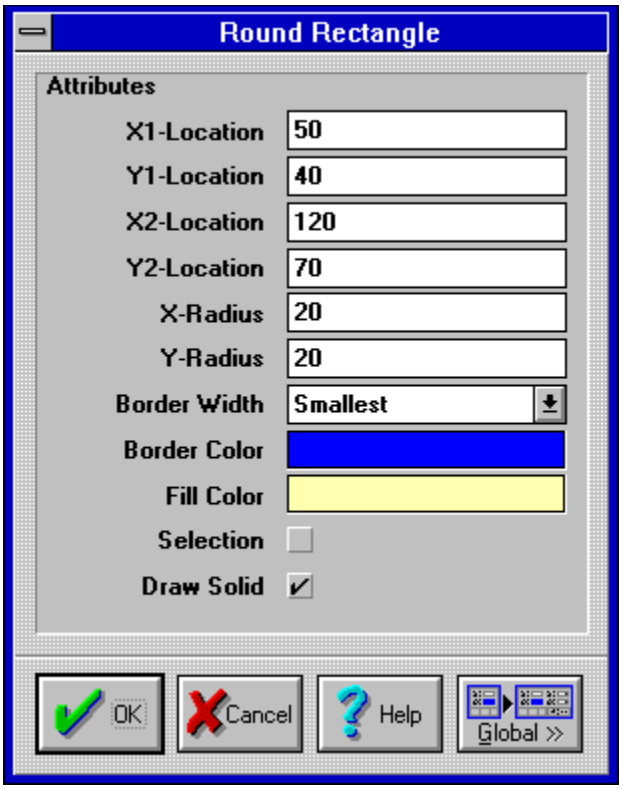
See also

Place Elliptical Arc
Ellipse

Round Rectangle

Overview Rounded rectangles are graphical objects with a user-defined line width, start/end coordinates, and fill color. Rounded rectangles are used for building library components, such as defining the component body or adding graphical symbols.





Fields

X1-Location The left hand edge reference point coordinate of the rounded rectangle, along the x-axis.

Y1-Location The bottom edge reference point coordinate of the rounded rectangle, along the y-axis.

X2-Location The right hand edge reference point coordinate of the rounded rectangle, along the x-axis.

Y2-Location The top edge reference point coordinate of the rounded rectangle, along the y-axis.

Schematic Library Editor allows you to change the horizontal corner radius of rounded rectangles. The corner radius can be any integer (whole number) value. A value of 1 = 10mil = .254mm.

Schematic Library Editor allows you to change the vertical corner radius of rounded rectangles. The corner radius can be any integer (whole number) value. A value of 1 = 10mil = .254mm.

Border Width Specifies the border outline thickness of Rounded Rectangles. There are four selectable border widths; smallest, small, medium and large.

Border Color The surrounding border of Rounded Rectangles can be assigned a color. To assign a new color to the border, click in the color box to open the Color Selector dialog box. Schematic Library Editor will display all the available colors that your graphics card and monitor supports.

Fill Color The inside area of Rounded Rectangles can be assigned a color. To assign a new color to the inside area, click in the color box to open the Color Selector dialog box. Schematic Library Editor will display all the available colors that your graphics card and monitor supports.

Selection Toggle the selection state of Rounded Rectangles by turning this option on or off. If this option is on, the Rounded Rectangle will be outlined in the selection color defined in the Options Preferences dialog box.

Selections are generally made three ways; using the option in a dialog box, using SHIFT+LEFT MOUSE to add (or remove) individual objects to the current selection and by using the Edit Select and De-select commands to define a selection group.

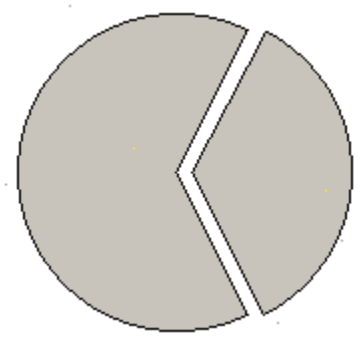
Draw Solid Schematic Library Editor allows you to turn the inside area color of Round Rectangles on or off. If this option is on, the inside area of the Rounded Rectangle will be displayed in the fill color. If this option is off, then only the Rounded Rectangles border will be shown.

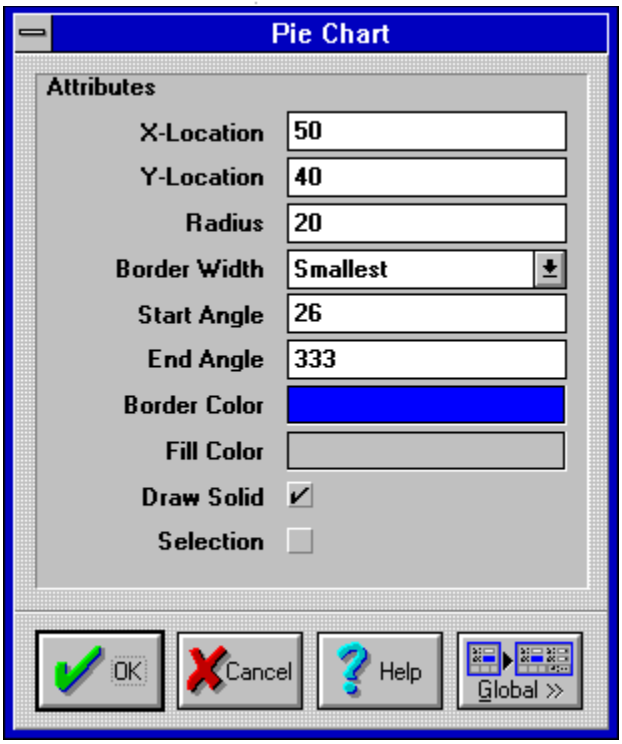
See also

Place Round Rectangle

Pie Chart

Overview Pie Charts are circular graphic objects with a user-defined line width, radius, starting angle and ending angle. Pie Charts are used for building library components, such as defining the component body or adding graphical symbols.





Fields

X-Location The reference point coordinate of the Pie object, along the x-axis.

Y-Location The reference point coordinate of the Pie object along the y-axis.

Radius Schematic Library Editor allows you to change the radius of Pie objects. The radius can be any integer (whole number) value. A value of 1 = 10mil = .254mm.

Start Angle Schematic Library Editor allows you to change the start angle of Pie objects. The start angle can be any degree from 1 to 360, (Real number).

End Angle Schematic Library Editor allows you to change the end angle of Pie symbols. The end angle can be any degree from 1 to 360, (Real number).

Border Color The surrounding border of Pie objects can be assigned a color. To assign a new color to

the border, click in the color box to open the Color Selector dialog box. Schematic Library Editor will display all the available colors that your graphics card and monitor supports.

Fill Color The inside area of Pie objects can be assigned a color. To assign a new color to the inside area, click in the color box to open the Color Selector dialog box. Schematic Library Editor will display all the available colors that your graphics card and monitor supports.

Draw Solid Schematic Library Editor allows you to turn the inside area color of Pie objects on or off. If this option is on, the inside area of the Pie will be displayed in the fill color. If this option is off, then only the Pie objects border will be shown.

Selection Toggle the selection state of Pie objects by turning this option on or off. If this option is on, the Pie will be outlined in the selection color defined in the Options Preferences dialog box.

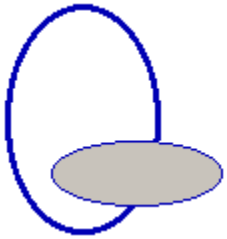
Selections are generally made three ways; using the option in a dialog box, using SHIFT+LEFT MOUSE to add (or remove) individual objects to the current selection and by using the Edit Select and De-select commands to define a selection group.

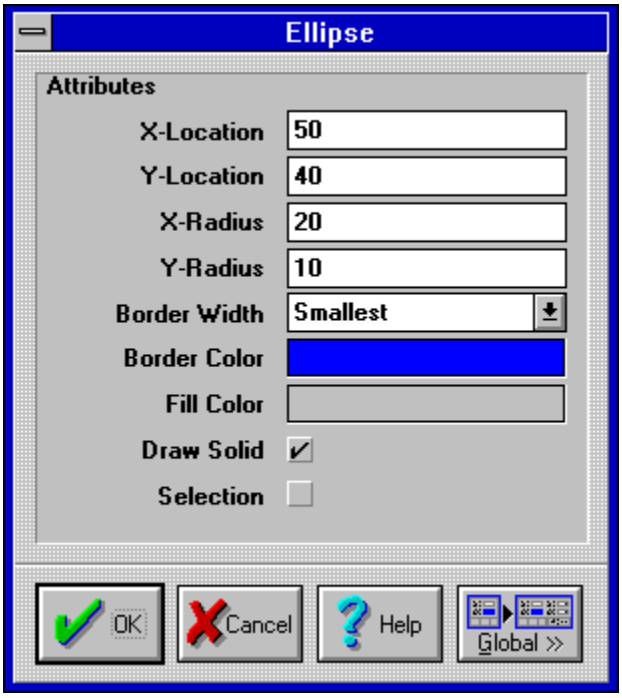
See also

Place Pie Charts

Ellipse

Overview Ellipses are graphical objects with a user-defined line width, x and y radius, and fill color. Ellipses are used for building library components, such as defining the component body or adding graphical symbols.





Fields

X-Location The reference point coordinate of an ellipse, along the x-axis.

Y-Location The reference point coordinate of the ellipse, along the y-axis.

X-Radius Schematic Library Editor allows you to change the horizontal width of Ellipses. The x-radius can be any integer (whole number) value. A value of 1 = 10mil = .254mm.

Y-Radius Schematic Library Editor allows you to change the vertical height of ellipse objects. The y-radius can be any integer (whole number) value. A value of 1 = 10mil = .254mm.

Border Width Specifies the border outline thickness of Ellipses. There are four selectable border widths; smallest, small, medium and large.

Border Color The surrounding border of Ellipses can be assigned a color. To assign a new color to the border, click in the color box to open the Color Selector dialog box. Schematic Library Editor will display all the available colors that your graphics card and monitor supports.

Fill Color The inside area of Ellipses can be assigned a color. To assign a new color to the inside area, click in the color box to open the Color Selector dialog box. Schematic Library

Editor will display all the available colors that your graphics card and monitor supports.

Draw Solid Schematic Library Editor allows you to turn the inside area color of Ellipses on or off. If this option is on, the inside area of the Ellipse will be displayed in the fill color. If this option is off, then only the Ellipses border will be shown.

Selection Toggle the selection state of Ellipses by turning this option on or off. If this option is on, the Ellipse will be outlined in the selection color defined in the Options Preferences dialog box.

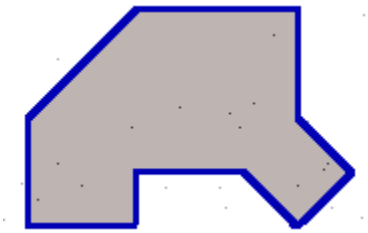
Selections are generally made three ways; using the option in a dialog box, using SHIFT+LEFT MOUSE to add (or remove) individual objects to the current selection and by using the Edit Select and De-select commands to define a selection group.

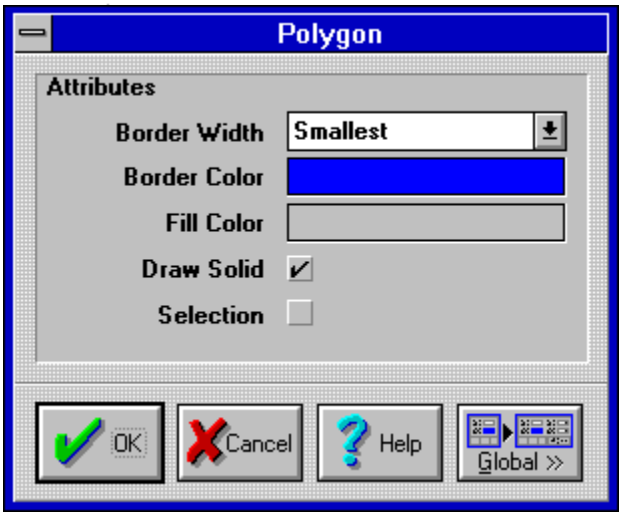
See also

Place Ellipse
Elliptical Arc

Polygon

Overview Polygons are graphical objects with a user-definable line width, vertex coordinates, and fill color. Each vertex of a polygon can be independently moved. Polygons are used for building library components, such as defining the component body or adding graphical symbols.





Fields

Border Width Specifies the border outline thickness of Polygons. There are four selectable border widths; smallest, small, medium and large.

Border Color The surrounding border of Polygons can be assigned a color. To assign a new color to the border, click in the color box to open the Color Selector dialog box. Schematic Library Editor will display all the available colors that your graphics card and monitor supports.

Fill Color The inside area of Polygons can be assigned a color. To assign a new color to the inside area, click in the color box to open the Color Selector dialog box. Schematic Library Editor will display all the available colors that your graphics card and monitor supports.

Draw Solid Schematic Library Editor allows you to turn the inside area color of Polygons on or off. If this option is on, the inside area of the Polygon will be displayed in the fill color. If this option is off, then only the Polygons border will be shown.

Selection Toggle the selection state of Polygon by turning this option on or off. If this option is on, the Polygon will be outlined in the selection color defined in the Options Preferences dialog box.

Selections are generally made three ways; using the option in a dialog box, using SHIFT+LEFT MOUSE to add (or remove) individual objects to the current selection and by

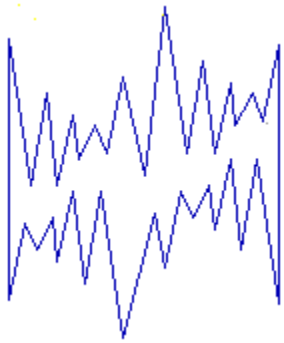
using the Edit Select and De-select commands to define a selection group.

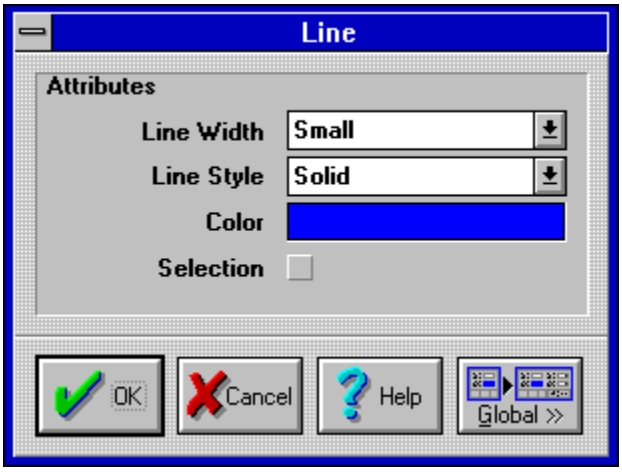
See also

Place Polygon

Line

Overview Lines are graphical objects with any number of joined segments and a user defined width, start and end point, vertices and color. Each vertex of a line can be independently moved. Lines are used for building library components, such as defining component outlines or adding graphical symbols.





Fields

Line Width Specifies the line thickness of Polyline segments. There are four selectable Polyline widths; smallest, small, medium and large.

Line Style Specifies the line display type of Polyline segments. There are three selectable Polyline styles; solid, dashed and dotted. The dashed and dotted styles only apply to 'smallest' width Polylines.

Color Polylines can be assigned a color. To assign a new color to the Polyline, click in the color box to open the Color Selector dialog box. Schematic Library Editor will display all the available colors that your graphics card and monitor supports.

Selection Toggle the selection state of Polylines by turning this option on or off. If this option is enabled, the Polyline will be outlined in the selection color defined in the Options Preferences dialog box.

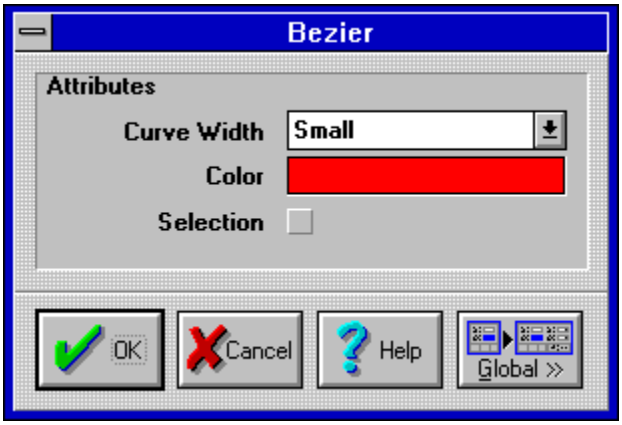
Selections are generally made three ways; using the option in a dialog box, using SHIFT+LEFT MOUSE to add (or remove) individual objects to the current selection and by using the Edit Select and De-select commands to define a selection group.

See also

Place Line

Bezier

Overview Beziers are curved line segments, used for building library components, such as defining component outlines or adding graphical symbols.



Fields

Curve Width Specifies the line thickness of the Bezier curves. There are four selectable line widths; smallest, small, medium and large.

Color Beziers can be assigned a color. To assign a new color to the lines and curves of the Bezier, click in the color box to open the Color Selector dialog box. Schematic Library Editor will display all the available colors that your graphics card and monitor supports.

Selection Toggle the selection state of Bezier curves by turning this option on or off. If this option is on, the Bezier curve will be outlined in the selection color defined in the Options Preferences dialog box.

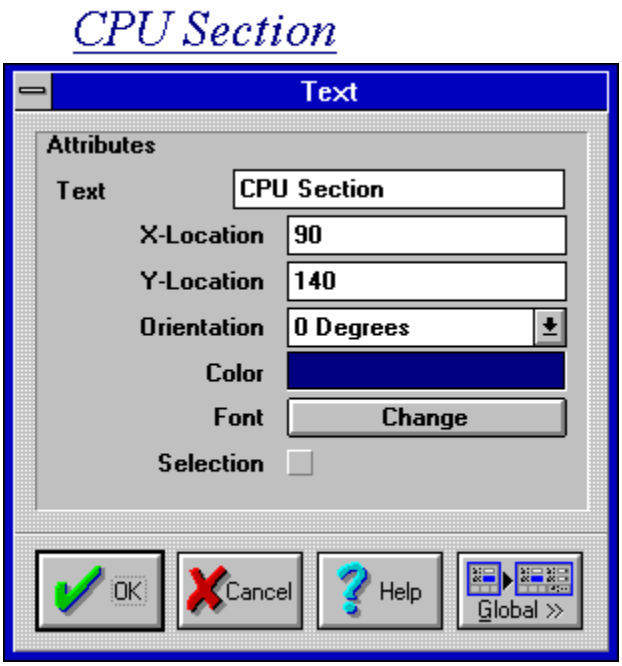
Selections are generally made three ways; using the option in a dialog box, using SHIFT+LEFT MOUSE to add (or remove) individual objects to the current selection and by using the Edit Select and De-select commands to define a selection group.

See also

Place Bezier

Text

Overview Single line free text is used to place notes or comments around a library component.



Fields

Text Free Text support up to 255 characters, including spaces.

X-Location The reference point coordinate of the free text, along the x-axis.

Y-Location The reference point coordinate of the free text, along the y-axis.

Orientation Text can be rotated through 360 degrees using 90 degree increments. There are four selectable orientations, 0 degrees, 90 degrees, 180 degrees and 270 degrees. While moving free text, you can rotate it around the cursor by pressing the spacebar.

Color Free text can be assigned a color. To assign a new color to the text, click in the color box to open the Color Selector dialog box. Schematic Library Editor will display all the available colors that your graphics card and monitor supports.

Font Free text support formatting, such as, True Type fonts, definable character sizes, italic, bold, underline, and strikeout styles.

Selection Toggle the selection state of the free text by turning this option on or off. If this option is on, the free text will be outlined in the selection color defined in the Options Preferences dialog box.

Selections are generally made three ways; using the option in a dialog box, using SHIFT+LEFT MOUSE to add (or remove) individual objects to the current selection and by using the Edit Select and De-select commands to define a selection group.

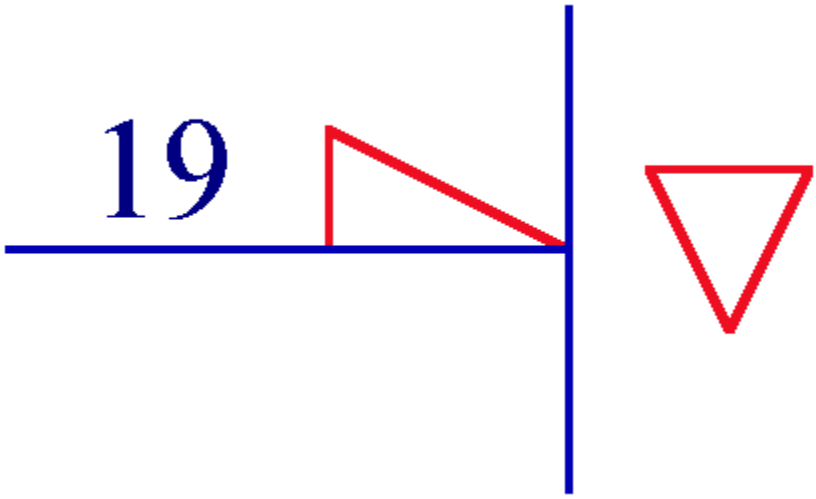
See also

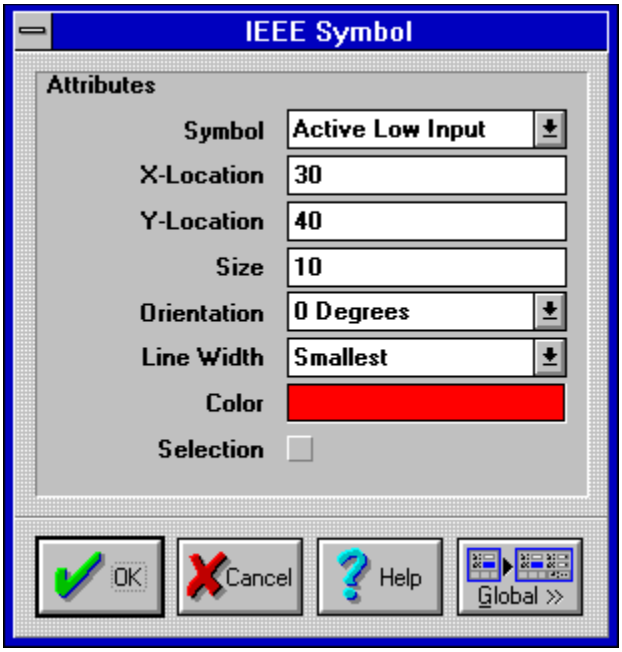
Place Text

IEEE Symbol

Overview IEEE Symbols are graphic qualifying symbols used for representing logic functions or devices. These symbols enable users to understand the logic characteristics of these functions or devices without requiring specific knowledge of their internal characteristics. The symbols can show the relationship of each input of a digital logic circuit to each output without showing explicitly the internal logic.

An IEEE symbol comprises an outline or a combination of outlines together with one or more qualifying symbols. Input lines are placed on the left and output lines are placed on the right. When an exception is made to that convention, the direction of signal flow is indicated by an arrow.





Fields

Symbol There are over thirty different qualifying symbols supported by Schematic Library Editor. The symbols are graphic representations only and have no electrical properties.

Dot: Logic negation at input (external 0 produces internal 1) or logic negation at output (internal 1 produces external 0).

Right Left Signal Flow: Signal flow from right to left. If not otherwise indicated, signal flow is from left to right.

Clock: Dynamic internal connection. Transition from 0 to 1 on left produces transitory 1 state on right.

Active Low Input: The L-level on the input produces the internal 1-state. Note: This qualifying symbol is designated the polarity symbol and shall point in the direction of signal flow.

Analog Signal In: Input for analog signals, on a digital symbol.

Not Logic Connection: Non logic connection. A label inside the symbol will usually define the nature of this pin. This symbol may be used to indicate a connection that does not carry any logic information, for example, a power supply connection.

Postponed Output: Postponed output. The change of the internal state of this output is postponed until the input signal that initiates the change returns to its initial external logic state or logic level.

Open Collector: Open-circuit output (L-type) (for example, NPN open-collector, PNP open-emitter, N-channel open-drain, or P-channel open-source). When not in its external high-impedance state, this type of output produces a relatively low-impedance L-level.

HiZ: 3-state output. This output can take on a third external state, which is a high-impedance state, having no logical significance.

High Current: Output with more than usual output capability (symbol is oriented in direction of signal flow).

Pulse: Monostable Elements.

Delay: Delay Elements.

Group Line: Line grouping for inputs and outputs. This symbol indicates that two or more terminals are needed to implement a single logic input/output.

Group Binary: Symbol used for binary grouping.

Active Low Output: Active-low output in the case of right-to-left signal flow.

Open Collector pull-up: Passive-pull-up output. Is similar to NPN open-collector output but is supplemented with a built-in passive pull-up.

Open Emitter: Open-circuit output (H-type) (for example, PNP open-collector, NPN open-emitter, P-channel open-drain, or N-channel open-source) When not in its external high-impedance state, this type of output produces a relatively low-impedance H-level.

Open Emitter pull-up: Passive-pull-down output. Is similar to NPN open-emitter output but is supplemented with a built-in passive pull-down.

Digital Signal In: Input for digital signals, on an analog symbol.

Input Output: Bi-directional signal flow. This symbol is used to indicate a single line that serves as both an input and a output.

Shift Left: Shifting input of a register from right to left.

Sigma: Adder.

Schmitt: Input with hysteresis, Bithreshold input.

Shift Right: Shifting input of a register from left to right.

X-Location The reference point coordinate of the IEEE symbol, along the x-axis.

Y-Location The reference point coordinate of an IEEE symbol, along the y-axis.

Size Specifies the size of IEEE symbol. There are four selectable sizes; smallest, small, medium and large.

Orientation IEEE symbols can be rotated through 360 degrees using 90 degree increments. There are four selectable orientations, 0 degrees, 90 degrees, 180 degrees and 270 degrees.

Line Width Specifies the line thickness of the IEEE symbol. There are four selectable line widths; smallest, small, medium and large.

Color IEEE symbols can be assigned with any color. To assign a new color to the IEEE symbol, click in the color box to open the Color Selector dialog box. Schematic Library Editor will display all the available colors that your graphics card and monitor supports.

Selection Toggle the selection state of IEEE symbols by turning this option on or off. If this option is on, the IEEE symbol will be outlined in the selection color defined in the Options Preferences dialog box.

Selections are generally made three ways; using this dialog box, using SHIFT+LEFT MOUSE to add (or remove) individual objects to the current selection and by using the Edit Select and De-select commands to define a selection group.

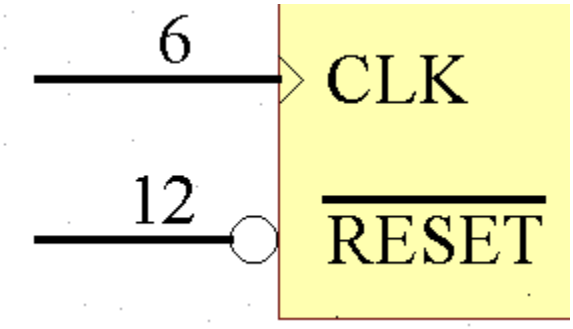
Comments While moving an IEEE symbol, you can rotate it 90 degrees around the cursor and flip it along its x or y axis by using the fs16 SPACEBAR, X and Y keys. Press TAB to manually change the default values of the symbol by typing new values directly into a dialog box.

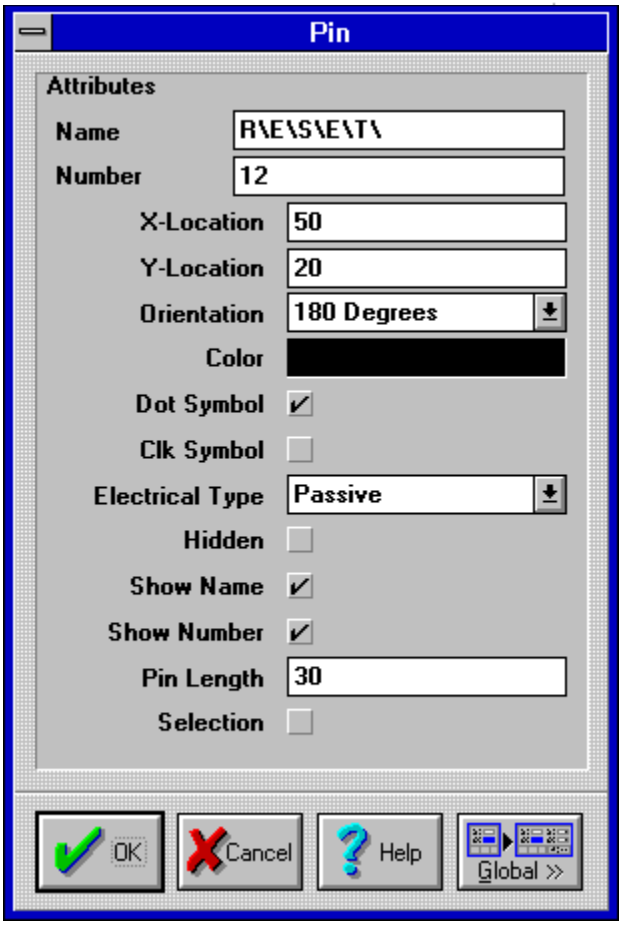
See also

Place IEEE Symbol

Pin

Overview Pins are special objects that have electrical characteristics and are used to direct signals into and out of Parts. Pins have user defined name and number, electrical type, length, color, orientation and other display options.





Fields

Name Pin names support up to 255 characters of text including spaces. The name is positioned to the left or to the right of the pin depending upon its orientation.

An inversion bar may be placed over the pin name characters by typing a backslash '\' after each character. For example, to place an inversion bar over the pin name 'ENABLE', type 'E\N\A\B\L\E\'. To place the inversion bar over the first two characters, type 'E\N\ABLE'.

Number Pin numbers support up to 255 characters of text including spaces. The number is positioned along the top of the pin. The pin number will automatically increment if the first or last characters are numbers. For example, an initial pin number of 1 will automatically increment to 2, 3, 4... and so on. A number of 2A will increment 3A, 4A, 5A, and a number of A10 will increment A11, A12, A13.

X-Location The reference point coordinate of the pin , along the x-axis.

Y-Location The reference point coordinate of the pin , along the y-axis.

Orientation Pins can be rotated through 360 degrees using 90 degree increments. There are four selectable orientations, 0 degrees, 90 degrees, 180 degrees and 270 degrees.

Color Pins can be assigned a color. To assign a new color to the Pin, click in the color box to open the Color Selector dialog box. Schematic Library Editor will display all the available colors that your graphics card and monitor supports.

Dot Symbol If this option is enabled, a small circular inversion dot will be displayed at the end of the pin.

Clk Symbol If this option is enabled, a small triangular clock identifier will be displayed at the end of the pin.

Electrical Type The electrical properties of Pins. The Electrical Rules Check recognizes Pin types and reports violations. There are eight electrical types that can be assigned to Pins:

Input - Specifies that the pin is to be used to direct a signal into a Part.

I/O - Specifies that the pin is bi-directional and can either be used as an input or an output pin.

Output - Specifies that the pin is to be used to direct a signal out of a Part.

Open Collector - Specifies that the pin is to be used in an open-collector gate. Open-collector gates are used in three major applications: driving a lamp or relay, performing wired logic, and for the construction of a common-bus system.

Passive - Specifies that the pin has no driving power source.

HiZ - A three-state pin exhibits three output states, a low-level state, a high-level state and an open circuit or high-impedance state.

Open Emitter - Specifies that the pin is to be used in an open-emitter gate.

Power - Specifies that the pin is connected to a power source or to ground.

Hidden If this option is enabled, the pin is hidden and is not displayed on the workspace. The pin also does not appear when placed on the Schematic Library Editor drawing workspace.

Show Name If this option is enabled, the pin name will be displayed and positioned to the left or to the right of the pin depending upon its orientation.

Show Number If this option is enabled, the pin number will be displayed and positioned along the top of the pin.

Pin Length Schematic Library Editor allows you to change the length of Pins. The length can be any integer (whole number) value. A value of 1 = 10mil = .254mm. The default is 30.

Selection Toggle the selection state of Pins by turning this option on or off. If this option is on, the Pin will be outlined in the selection color defined in the Options Preferences dialog box.

Selections are generally made three ways; using the option in a dialog box, using SHIFT+LEFT MOUSE to add (or remove) individual objects to the current selection and by using the Edit Select and De-select commands to define a selection group.

See also

Place Pin
Toggle Hidden Pins

MainMenu

File	Nested Menu
Edit	Nested Menu
Place	Nested Menu
Component	Nested Menu
Tools	Nested Menu
Options	Nested Menu
Zoom	Nested Menu
Info	Nested Menu
Window	Nested Menu
Help	Nested Menu

File

New	Create a new and empty component library file
Open...	Open and load a component library file
Merge...	Merge components from another library to current library file
Close	Close the current window
Save	Save current library with same file name
Save As...	Save current component library with new file name
Save All	Save all loaded component library files
Setup Printer...	Printer and page setup
Print	Print the current document with the last set printer options
Report	Nested Menu
Update Schematic	Update components in all opened schematic worksheet files
Exit	Quit from Schematic Library Editor

Report

Component	Display component information
Library	Display library statistics
Component Rule Check	Validate changes to current library

Edit

Undo	Undo previous command
Redo	Redo previous undo command
Cut	Copy selected objects to clipboard and remove from workspace
Copy	Copy all selected objects to the clipboard
Paste	Place clipboard contents onto current library workspace
Paste Array...	Define an array placement of clipboard contents
Clear	Delete all selected objects from the current library workspace
Find Text...	Search for and jump to a text string on the workspace.
Replace Text...	Search for and replace text strings
Find Next	Search for and jump to the next matching text string
Select	Nested Menu
DeSelect	Nested Menu
Toggle Selection	Toggle selection-state of objects
Delete	Select and delete objects on library workspace
Change	Select and use dialog to change objects
Move	Nested Menu
Align	Nested Menu
Jump	Nested Menu

Select

Inside Area	Select all objects inside an area
Outside Area	Select all objects outside an area
All	Select everything on the current library workspace

DeSelect

Inside Area	De-Select all objects inside an area
Outside Area	De-Select all objects outside an area
All	De-Select all selected objects

Move

Move	Select and move objects on the library workspace
Move Selection	Move selected objects to another area of the library workspace
Move To Front	Move and place an object in the front of all other objects
Bring To Front	Bring an object graphically to the front of all other objects
Send To Back	Send an object to the back of all other objects
Bring To Front Of	Bring an object to the front of another object
Send To Back Of	Send an object to the back of another object

Align

Align...	Align selected objects using alignment dialog box
Align left	Align selected objects on the left side of their bounding rectangle
Align Right	Align selected objects on the right side of their bounding rectangle
Center Horizontal	Center objects around the vertical center line of the bounding rectangle
Distribute Horizontally+Shift	Distribute equally the selected objects along the horizontal axis
Align Top	Align selected objects on the top side of their bounding rectangle
Align Bottom	Align selected objects on the bottom side of their bounding rectangle
Center Vertical	Center objects around the vertical center line of the bounding rectangle
Distribute Vertically+Shift	Distribute equally the selected objects along the vertical axis

Jump

Origin	Jump to the origin of the library workspace (lower left)
New Location...	Type in and jump to a new location on the worksheet

Place

IEEE Symbols	Nested Menu
Pins	Place electrical pin on the library workspace
Junction Symbol	Place a Junction symbol on the library workspace
Arcs	Place graphical arcs on the library workspace
Elliptical Arcs	Place elliptical arcs on the library workspace
Ellipses	Place elliptical shapes on the library workspace
Pie Charts	Place pie charts on the library workspace
Line	Place graphical lines on the library workspace
Rectangle	Place rectangles on the library workspace
Round Rectangle	Place round rectangles on the library workspace
Polygons	Place graphical polygon shapes on the library workspace
Beziers	Place curved lines on the library workspace
Text	Place single line text on the library workspace
Graphic...	Place PCX/BMP/GIF/TIFF/WMF/EPS graphical images on the library workspace

IEEE Symbols

Dot	Place IEEE Dot symbol on the library workspace
Right Left Signal Flow	Place IEEE Right Left Signal Flow symbol on the library workspace
Clock	Place IEEE Clock symbol on the library workspace
Active Low Input	Place IEEE Active Low Input symbol on the library workspace
Analog Signal In	Place IEEE Analog Signal In symbol on the library workspace
Not Logic Connection	Place IEEE Not Logic Connection symbol on the library workspace
Postponed Output	Place IEEE Postponed Output symbol on the library workspace
Open Collector	Place IEEE Open Collector symbol on the library workspace
HiZ	Place IEEE Hiz symbol on the library workspace
High Current	Place IEEE High Current symbol on the library workspace
Pulse	Place IEEE Pulse symbol on the library workspace
Delay	Place IEEE Delay symbol on the library workspace
Group Line	Place IEEE Group Line symbol on the library workspace
Group Binary	Place IEEE Group Binary symbol on the library workspace
Active Low Output	Place IEEE Active Low Output symbol on the library workspace
Pi Symbol	Place IEEE Pi symbol on the library workspace
Greater Equal	Place IEEE Greater Equal symbol on the library workspace
Open Collector PullUp	Place IEEE Open Collector Pull Up symbol on the library workspace
Open Emitter	Place IEEE Open Emitter symbol on the library workspace
Open Emitter PullUp	Place IEEE Open Emitter Pull Up symbol on the library workspace
Digital Signal In	Place IEEE Digital Signal In symbol on the library workspace
Invertor	Place IEEE Invertor symbol on the library workspace
Or Gate	Place IEEE Or Gate symbol on the library workspace
Input Output	Place IEEE Input Output symbol on the library workspace
And Gate	Place IEEE And Gate symbol on the library workspace
Xor Gate	Place IEEE Xor Gate symbol on the library workspace
Shift Left	Place IEEE Shift Left symbol on the library workspace
Less Equal	Place IEEE Less Equal symbol on the library workspace
Sigma	Place IEEE Sigma symbol on the library workspace
Schmitt	Place IEEE Schmitt symbol on the library workspace
Shift Right	Place IEEE Shift Right symbol on the library workspace

Component

New Component	Add a new component to library
Remove Component	Remove component from the current library
Rename Component...	Change the name of the current component
Remove Component Name	Remove component name from group
Add Component Name...	Add component name to group
Copy Component...	Copy component from current library to another library file
Move Component...	Move component from current library to another library file
New Part	Add a new part to the currently displayed component
Remove Part	Remove a part from the currently displayed component
Component	Nested Menu
Show Normal	Display component in standard ANSI notation
Show Demorgan	Command Summary
Show IEEE	Command Summary
Description...	Change component description fields
Remove Duplicates	Remove duplicate components from library

Component

Next Part	Show the next part of the currently displayed component
Prev Part	Show the previous part of the currently displayed component
Next Component	Show the next component in the library
Prev Component	Show the previous component in the library
First Component	Show the first component in the library
Last Component	Show the last component in the library

Tools

Schematic Capture	Launch and Switch to the Schematic Sheet Editor
Windows Tools	Nested Menu
User Tools	Nested Menu
Setup...	Setup user specific Programs to use with Tools Menu

Windows Tools

File Manager	Run the Windows File Manager Program
Control Panel	Run the Windows Control Panel Program
Windows Setup	Run the Windows Setup Program
Calculator	Run the Windows Calculator Program
Clock	Run the Windows Clock Program
Notepad	Run the Windows Notepad program
Text Editor	Run the user specified text editor. (Defaults To Notepad.Exe)
Picture Editor	Run the user specified Paint/Draw program (Default is Windows Paint)
CSV Editor	Run the user specified program for editing CSV files

User Tools

User Program 1	Run user specified Program 1
User Program 2	Run user specified Program 2
User Program 3	Run user specified Program 3
User Program 4	Run user specified Program 4

Options

Preferences...	Setup System Preferences
Workspace...	Setup options for current workspace
Hot Keys...	Setup Hot Key assignments
Auto-Pan...	Setup Autopan options
Memory Monitor...	Setup low memory/resources warning thresholds.
Status Bar	Turn the Status Line on or off
Command Status Bar	Turn the Command Status Line on or off
Scroll Bars	Turn the scroll bars on or off
Main Panel	Turn Main Panel on or off
Main Toolbar	Turn the Main Toolbar on or off
IEEE Toolbar	Turn the IEEE Toolbar on or off
Drawing Toolbar	Turn the Drawing Toolbar on or off
Visible Grid	Turn the visible grid on or off
Snap Grid	Turn the snap grid on or off
Show Hidden Pins	Show/Hide hidden pins in component
Open...	Open and load an environment configuration file
Save As...	Save current environment configuration to a file

Zoom

Window	Select a rectangular area of the workspace and fit that area in the window
Point	Select a rectangular area of the workspace and fit that area in the window
50%	Set zoom scale to 0.5x
100%	Set zoom scale to 1x (normal magnification)
200%	Set zoom scale to 2x
400%	Set zoom scale to 4x (highest magnification)
In	Show less of the current workspace (higher magnification)
Out	Show more of the current workspace (lower magnification)
Pan	Re-Center the screen around the cursor
Redraw	Update the screen display
All	Fit all objects on the current workspace in the window
Workspace	Show the entire workspace

Info

System Status... Display system information

Window

Tile	Tile all open schematic windows
Cascade	Cascade all open schematic windows
Arrange Icons	Arrange all minimized open library document windows
Close All	Close all open schematic windows

Help

Contents	Help system topic index
Using Help	Information about the Windows Help System
Basic Concepts	Basic information about Schematic Library Editor
Commands	Help information organized by menu structure
Printing	Information about generating hard copy output
Reference	Schematic Library Editor Reference
About...	Display the version number and copyright of Schematic Library Editor

Hot Keys

E	**Popup Edit Menu** Popup the Edit menu
C	**Popup Component Menu** Popup the Component menu
M	**Popup Move Menu** Popup the Edit Move menu
P	**Popup Place Menu** Popup the Place menu
J	**Popup Jump Menu** Popup the Edit Jump menu
S	**Popup Select Menu** Popup the Edit Select menu
X	**Popup De-Select Menu** Popup the Edit De-Select menu
Z	**Popup Zoom Menu** Popup the Zoom menu
O	**Popup Options Menu** Popup the Options menu
A	**Popup Alignment Menu** Popup the Alignment menu
H	**Popup Help Menu** Popup the Help menu
F	**Popup File Menu** Popup the File menu
I	**Popup Info Menu** Popup the Info menu
W	**Popup Window Menu** Popup the Window menu
F1	**Help Index** Help system topic index
Shift F4	**Window Tile** Tile all open schematic windows
Shift F5	**Window Cascade** Cascade all open schematic windows
Alt BackSpace	**Undo** Undo previous command
Ctrl BackSpace	**Redo** Redo previous undo command
Shift Delete	**Cut** Copy selected objects to clipboard and remove from workspace
Ctrl Insert	**Copy** Copy all selected objects to the clipboard
Shift Insert	**Paste** Place clipboard contents onto current library workspace
Ctrl Delete	**Clear** Delete all selected objects from the current library workspace
Shift MouseLeft	**Toggle Single Object** Toggle selection-state of objects
Delete	**Delete Single Object** Delete object that currently has the focus
MouseLeft	**Change Object Graphically or Set Focus** Select and change objects on a library component
MouseLeftHold	**Change Object Graphically or Move** Select and change objects on a library component
MouseDblLeft	**Change Single Object** Use dialog to change object under the cursor
Ctrl MouseLeft	**Move Single Object** Select and move objects on the library workspace
Ctrl PgDn	**Zoom All** Fit all objects on the current workspace in the window
PgUp	**Zoom In** Show less of the current workspace (higher magnification)
PgDn	**Zoom Out** Show more of the current workspace (lower magnification)
Home	**Zoom Pan** Re-Center the screen around the cursor
End	**Screen Redraw** Update the screen display
Ctrl 4	**Zoom 400%** Set zoom scale to 4x (highest magnification)
Ctrl 2	**Zoom 200%** Set zoom scale to 2x
Ctrl 1	**Zoom 100%** Set zoom scale to 1x (normal magnification)
Ctrl 5	**Zoom 50%** Set zoom scale to 0.5x
Shift Up	**Shift Cursor Up** Move the cursor up 10 snap grid points
Shift Down	**Shift Cursor Down** Move the cursor down 10 snap grid points
Shift Left	**Shift Cursor Left** Move the cursor up 10 snap grid points
Shift Right	**Shift Cursor Right** Move the cursor right 10 snap grid points

Up	**Cursor Up** Move the cursor up one snap grid point
Down	**Cursor Down** Move the cursor down one snap grid point
Left	**Cursor Left** Move the cursor left one snap grid point
Right	**Cursor Right** Move the cursor right one snap grid point
Ctrl F	**Find Text** Search for and jump to a text string on the workspace.
F3	**Find Next Text** Search for and jump to the next matching text string
Ctrl G	**Find and Replace Text** Search for and replace text strings
Ctrl L	**Align objects on left** Align selected objects on the left side of their bounding rectangle
Ctrl R	**Align objects on right** Align selected objects on the right side of their bounding rectangle
Ctrl H	**Center objects around horizontal axis** Center objects around the vertical center line of the bounding rectangle
Ctrl Shift H	**Distribute equally along horizontal axis** Distribute equally the selected objects along the horizontal axis
Ctrl T	**Align objects on top** Align selected objects on the top side of their bounding rectangle
Ctrl B	**Align objects on bottom** Align selected objects on the bottom side of their bounding rectangle
Ctrl V	**Center objects around vertical axis** Center objects around the vertical center line of the bounding rectangle
Ctrl Shift V	**Distribute equally along vertical axis** Distribute equally the selected objects along the vertical axis

Tool Bars

Tools

IEEE

Library Tools

Tools

Toggle Main Panel Turn Main Panel on or off

File Open Open and load a component library file

File Save Save current library with same file name

Setup Printer Printer and page setup

Zoom In Show less of the current workspace (higher magnification)

Zoom Out Show more of the current workspace (lower magnification)

Cut Copy selected objects to clipboard and remove from workspace

Paste Place clipboard contents onto current library workspace

Select Inside Area Select all objects inside an area

De-Select All De-Select all selected objects

Move Selection Move selected objects to another area of the library workspace

Run Schematic Capture Launch and Switch to the Schematic Sheet Editor

Toggle Drawing Toolbar Turn the Drawing Toolbar on or off

Toggle IEEE Toolbar Turn the IEEE Toolbar on or off

Undo Undo previous command

Redo Redo previous undo command

Help Index Help system topic index

IEEE

Place IEEE Dot Place IEEE Dot symbol on the library workspace

Place IEEE Right Left Signal Flow Place IEEE Right Left Signal Flow symbol on the library workspace

Place IEEE Clock Place IEEE Clock symbol on the library workspace

Place IEEE Active Low Input Place IEEE Active Low Input symbol on the library workspace

Place IEEE Analog Signal In Place IEEE Analog Signal In symbol on the library workspace

Place IEEE Not Logic Connection Place IEEE Not Logic Connection symbol on the library workspace

Place IEEE Postponed Output Place IEEE Postponed Output symbol on the library workspace

Place IEEE Open Collector Place IEEE Open Collector symbol on the library workspace

Place IEEE Hiz Place IEEE Hiz symbol on the library workspace

Place IEEE High Current Place IEEE High Current symbol on the library workspace

Place IEEE Pulse Place IEEE Pulse symbol on the library workspace

Place IEEE Delay Place IEEE Delay symbol on the library workspace

Place IEEE Group Line Place IEEE Group Line symbol on the library workspace

Place IEEE Group Binary Place IEEE Group Binary symbol on the library workspace

Place IEEE Active Low Output Place IEEE Active Low Output symbol on the library workspace

Place IEEE Pi Symbol Place IEEE Pi symbol on the library workspace

Place IEEE Greater Equal Place IEEE Greater Equal symbol on the library workspace

Place IEEE Open Collector Pull Up Place IEEE Open Collector Pull Up symbol on the library workspace

Place IEEE Open Emitter Place IEEE Open Emitter symbol on the library workspace

Place IEEE Open Emitter Pull Up Place IEEE Open Emitter Pull Up symbol on the library workspace

Place IEEE Digital Signal In Place IEEE Digital Signal In symbol on the library workspace

Place IEEE Invertor Place IEEE Invertor symbol on the library workspace

Place IEEE Input Output Place IEEE Input Output symbol on the library workspace

Place IEEE Shift Left Place IEEE Shift Left symbol on the library workspace

Place IEEE Less Equal Place IEEE Less Equal symbol on the library workspace

Place IEEE Sigma Place IEEE Sigma symbol on the library workspace

Place IEEE Schmitt Place IEEE Schmitt symbol on the library workspace

Place IEEE Shift Right Place IEEE Shift Right symbol on the library workspace

Library Tools

Place Line Place graphical lines on the library workspace

Place Bezier Place curved lines on the library workspace

Place Elliptical Arc Place elliptical arcs on the library workspace

Place Polygon Place graphical polygon shapes on the library workspace

Place Text Place single line text on the library workspace

New Component Add a new component to library

New Part Add a new part to the currently displayed component

Place Rectangle Place rectangles on the library workspace

Place Round Rectangle Place round rectangles on the library workspace

Place Ellipse Place elliptical shapes on the library workspace

Place Junction Symbol Place a Junction symbol on the library workspace

Place Graphic Place PCX/BMP/GIF/TIFF/WMF/EPS graphical images on the library workspace

Paste Array Define an array placement of clipboard contents

Place Pin Place electrical pin on the library workspace

Error Messages

Unrecognized File Format

This message occurs whenever a sheet file or library file Open command encounters an unrecognized format. Advanced Schematic will load Protel Schematic, Orcad STD III and STD IV sheet files automatically. The Library Editor will load Protel Advanced Schematic binary and OrCAD binary formats plus decompiled Protel (DOS) and OrCAD text format libraries. Other formats will generate this error message.

See also

Error Messages

Too Many Files Open

The number of files specified in the FILES= command in your CONFIG.SYS file has been exceded. Close one or more files or increase the number of files specified in CONFIG.SYS.

See also

Error Messages

Text not found

In the Text find and replace commands, the text that you specified was not found anywhere on the worksheet.

See also

Error Messages

Sector Not Found

Internal error. Contact Protel for assistance.

See also

Error Messages

Printing Aborted

You have pressed Cancel in the Print dialog box. Choose the Print command and re-print the job, if desired.

See also

Error Messages

Print Manager Aborted

You have closed the print manager while a job is printing. Choose the Print command and re-print the job.

See also

Error Messages

Path not found

You have specified a path that doesn't exist.

See also

Error Messages

Out of Memory

While placing items, loading a file or general editing, the program has run out of memory. Clear the Undo stack or close other applications to make more memory available. Alternatively exit and restart Windows. If none of these options work, then you will need to add additional memory to your system to perform this operation.

See also

Error Messages

Out Of Memory While Printing

The program has run out of memory. Clear the Undo stack or close other applications to make more memory available. Alternatively exit and restart Windows. If none of these options work, then you will need to add additional memory to your system to perform this operation.

See also

Error Messages

Out Of Disk Space While Printing

When printing to file, sufficient disk space must be available. Either select another drive or clear enough disk space for the print file and re-attempt printing.

See also

Error Messages

Orcad STDXTEND.EXE must be in the PATH to Load Orcad 32-bit Binary File

The DOS memory 32 bit memory utility STDXTEND.EXE is supplied with the OrCAD SDT 386-32 Bit system and is used by their 32 bit applications. Make sure that it is in your PATH.

See also

Error Messages

Orcad DECOMP.EXE (Renamed to DECOMP32.EXE) must be in the PATH to Load Orcad 32-bit Binary Library

The DOS application DECOMP.EXE is supplied with the OrCAD SDT 386-32 bit system. Copy this file to DECOMP32.EXE and make sure that it is in your PATH. This allows Advanced Schematic to load both 16 and 32 bit Orcad Libraries.

See also

Error Messages

Orcad 32To16.EXE must be in the PATH to Load Orcad 32-bit Binary Schematic

The DOS application 32To16.EXE is supplied with the OrCAD SDT 386-32 Bit system. Make sure that it is in your PATH. Advanced Schematic needs this utility to load Orcad 32 bit schematics.

See also

Error Messages

Not Enough Memory To Start Application

There is not memory and/or resource space to start the application

See also

Error Messages

No selection

Use the Editcommands or SHIFT+LEFT MOUSE to select some items.

See also

Error Messages

No selected pins

You have selected Information Selected Pins with no pins currently selected.

See also

Error Messages

No printer found

You must selected a windows printer in File Plot/Print before choosing Generate Prints. Use Window Control Panel to install printers.

See also

Error Messages

No more Space For Access Codes

Only 40 access codes can be added at one time

See also

Error Messages

No memory left

While placing items, loading a file or general editing, the program has run out of memory. Clear the Undo stack or close other applications to make more memory available. Alternatively exit and restart Windows. If none of these options work, then you will need to add some more memory.

See also

Error Messages

No Error Markers Found

The Jump to error markers command could not find any error markers in the current sheet. Use File:Reports:ERC to find errors.

See also

Error Messages

No Duplicate Components

Remove Duplicates command has not found any duplicate components in the library.

See also

Error Messages

Library Must Be Protel Advanced Schematic Format

The library file which you have tried to load, merge or Add to the libaray list, is not saved in a valid Protel or Orcad format. It may not be the right file (a Protel Schematic file for example) or it may have been damaged. If it is damaged, then you will have to resort to a back-up version.

See also

Error Messages

Invalid OrCAD library file format

The library file which you have tried to load or merge from (extension SRC) is not saved in a valid Orcad ASCII (decompiled) format. It may not be the right file (an Orcad file for example) or it may have been damaged. If this occurs, try loading the file in LIB (OrCAD binary) format. Advanced Schematic will perform the de-compilation automatically in this case. If the library has been succesfully Decompiled but still generates this error, then you should used the Orcad utility CONVERT.EXE to convert it to 16 bit before loading it into Library Editor.

See also

Error Messages

Invalid File Handle

Internal error. Contact Protel for assistance.

See also

Error Messages

Invalid File Access

Internal error. Contact Protel for assistance.

See also

Error Messages

Invalid Drive

The requested drive is not recognized by the system. Please specify the correct drive.

See also

Error Messages

Invalid Code, Please Re-Enter

You have entered an access code with an incorrect format. Correct format is four hexadecimal characters, followed by a hyphen (-) and four additional hexadecimal characters. Be sure that you type zero (0) and not the letter "O" where it appears in the code sequences.

See also

Error Messages

Incorrect Access Code For Schematic

You have attempted to run this module without having typed the correct access code into the Set Acces Codes dialog box. To open this dialog box, choose the Help About command and click Set Access Codes. Carefully type the correct code into the box for this module and press Test to verify each code. Be careful to type zero (0) and not the letter "O" where it appears in the code sequences.

See also

Error Messages

I/O Error Number :

Internal error. Note the error message/number and contact Protel for assistance.

See also

Error Messages

Hard Lock Device Not Found

You have attempted to run a protected Protel for Windows application without having first connected the external security device supplied with the system. Make sure that the device is properly connected to a parallel port and re-start Protel for Windows. If this is unsuccessful, the device may be either damaged or defective. In this case, contact your Protel representative for assistance. Warning: incorrect connection may damage the device -- make sure that the device is connected to a parallel, not serial port, with the "arrow" pointing toward the printer.

See also

Error Messages

General Error While Printing

Communication with the printer has been interrupted. Check the printer connections, cabling, etc. and retry.

See also

Error Messages

File Not Open

Internal error. Contact Protel for assistance.

See also

Error Messages

File not found

The file name selected could not be found.

See also

Error Messages

File is Read Only

You are trying to save to a file that is Read Only. Either change the files attributes or use Save As to write to a new file name.

See also

Error Messages

File Error :

Internal error. Note the error message/number and contact Protel for assistance.

See also

Error Messages

File Corruption Error

The Schematic worksheet file which you have tried to load from (extension ASC) contains text which does not conform to the Protel Schematic file format. This error indicates that the program recognized the file as a valid schematic file, but then while actually loading it, some part of it was invalid. If you have been editing the file manually (with a text editor) then see the manual for more information or contact Protel Technical support. Or , if the file is a foriegn format, OrCAD for example, it may contain elements that are not recognised by the Protel loader.

See also

Error Messages

File Access Denied

The file you are attempting to open is either currently in use or you are attempting to write to a read-only file.

See also

Error Messages

Feature Not Enabled In This Version

This feature is not available in the the currently installed version.

See also

Error Messages

Drive Write Error

The requested drive or file cannot be accessed. Make sure that the correct drive has been specified.

See also

Error Messages

Drive Seek Error

The requested drive or file cannot be accessed. Make sure that the correct drive has been specified.

See also

Error Messages

Drive Read Error

The requested drive or file cannot be accessed. Make sure that the correct drive has been specified.

See also

Error Messages

Drive Not Ready

The requested drive is not available or responding. Make sure that the correct drive has been specified.

See also

Error Messages

Disk Is Write Protected

The diskette is write protected. Remove or open the write protect tab.

See also

Error Messages

Cannot Switch To Target Application, Multiple Copies of Application Running

For inter-application communications, multiple target copies create ambiguities. This can occur from Place Component from the library editor, Edit component from Schematic and cross probing from any application.

See also

Error Messages

Using Schematic Library Editor Help

This on-line guide provides an introduction to the Advanced Schematic Library Editor application, including many of the features, key concepts and terminology used throughout the system. A separate on-line help system is maintained for the Advanced Schematic application. The guide is intended to provide the general information you need to get up and running with the system and to learn to use the basic features required to design a circuit, generate a netlist and print artwork. Detailed information can be found in your Advanced Schematic documentation, including the *Protel for Windows Environment Guide*, *Advanced Schematic User Guide*, *Schematic Editor Reference* and *Library Editor Reference*.

Help methods

There are three methods for using On-line Help.

1. You can run the Help application and switch to Help, as needed, using the Contents and Search facilities to browse the available topics.
2. A second, powerful way to use help is to use context sensitivity. By default the F1 key is linked to the Help system. When you are running Advanced Schematic, pressing F1 will open the relevent topic when you are in placement mode or when a menu is selected.
3. You can press the Help button in most dialog boxes. This will run context sensitive help for that dialog.

System overview
Assumptions made by this guide

System overview

Advanced Schematic includes two independent applications: the Schematic Editor and the Schematic Library Editor.

Advanced Schematic can generate single sheet, multiple sheet and fully hierarchical designs of virtually any size, limited only by the available memory and storage capacity of your system. Sheet sizes include A, B, C, D, E (or metric sizes A4-A0) plus user-defined sheets of up to 65 inches square. Standard component libraries include over 15,000 parts, with standard ANSI, DeMorgan and IEEE display options (where applicable).

Schematic Editor
Schematic Library Editor

Schematic Editor

The Schematic Editor is the primary Advanced Schematic application. This application allows the user to create, edit, check and print the sheet files that comprise a design project. All the tools and utilities needed to generate valid netlists, design reports and presentation quality schematic drawings are integrated into the Schematic Editor application.

Open and view as many sheets and design files as desired. Any number of Advanced Schematic sheets can be opened in their own windows, limited only by available memory. Run multiple Windows (and Windows-compatible DOS) applications. Switch between sheets, files and applications with a click of the mouse.

Schematic Library Editor

The Library Editor is an independent application used to create, edit and manage parts libraries and their components. You can run the Schematic Editor concurrently with the Library Editor with special links that allow you to move conveniently back and forth between the two applications. The Schematic Library Editor includes many of the features of the Schematic Editor application, plus specialized tools and features for component part creation and library management.

File, Netlist and output format options

Advanced Schematic loads OrCAD SDT and Protel-Schematic (DOS version 3.3) files, along with Advanced Schematic files. Translation of imported drawings is 100% and full support is provided for all Protel and OrCAD SDT design objects and functions. De-compiled OrCAD SDT libraries (and Protel-Schematic (DOS) libraries) can be translated into Advanced Schematic format, as well. A large number of netlist output formats are supported. Windows 3 features support for many output devices including dot matrix printers, PostScript printers and imagesetters, as well as plotters of many different types.

About editing

Attributes can be edited by double-clicking directly on the item to open a dialog box. In Advanced Schematic, changes can be globally applied across an entire design using specific conditions to define the targets. For example, when editing wires you can change the color or wire size or both attributes. These changes can be globally applied to other wires on the sheet, or to other open sheets. Similar global options are provided for components and other objects.

About Libraries

Advanced Schematic includes a powerful system for managing schematic component libraries. Any number of part libraries can be opened and accessed simultaneously while you create your drawings. Comprehensive standard manufacturer libraries are included with Protel for Windows. Components can be browsed and placed directly from the independent Schematic Library Editor application. Simultaneous multi-user library access is supported for network installations.

Array placement options

Linear array placement allows automated step-and-repeat placement of objects in the sheet. This can include individual objects or complex selections of objects. The users can specify the number of repeats and set pre-defined x and y offsets, and text increments.

See also

Using the clipboard
Place Array command

Windows support for printing and plotting

Dot matrix and laser printing, color printing, pen plotting and PostScript output are all controlled from a common Print command. Any device supported by Windows can be selected. Advanced Schematic graphical design tools support the production of presentation quality artwork.

Windows display options

Windows display options

Protel for Windows makes full use of all standard and 24 bit color graphics cards and monitors supported under Windows 3. On standard graphics adapters such as VGA, dithering can be used to simulate colors beyond the standard 20 Windows solid tones. Over 700 pre-defined standard colors on a special palette are provided, along with the ability to define custom colors. Protel for Windows makes full use of the Windows graphical environment, allowing the user to place TrueType fonts, place bitmap and vector graphics images and to assign colors independently to all display items.

Assumptions made by this guide

The documentation makes four assumptions about Advanced Schematic users:

1. That users have a sound knowledge of the principles and terminology of schematic design and capture;
2. That users are familiar with Windows 3 icons, menus, windows and using the mouse to make selections.
3. That users have a basic understanding about how Windows manages applications (programs and utilities) and documents (data files) to perform routine tasks such as starting applications, opening documents and saving work. If you are new to Windows, please start with your *Microsoft Windows 3.0 User's Guide*.
4. That users have a basic understanding of Microsoft DOS and its use of directories, file naming conventions, etc.

Protel for Windows Environment Guide

The Protel for Windows design system currently includes printed circuit board layout, PCB design automation and schematic design applications. The *Protel for Windows Environment Guide*, delivered with your Advanced Schematic package, provides an overview of the Protel for Windows design environment, including many of the features and concepts used throughout the various modules of the system. It provides step-by-step instructions for installing PfW software, some applicable Windows fundamentals and information regarding technical support and product upgrades.

Advanced Schematic User Guide

The *User Guide* is designed to guide the new user through the many features of Advanced Schematic. It includes general coverage of the basic principles used througout the Advanced Schematic system, including concepts and an introduction to Advanced Schematic objects, tools and processes.

Advanced Schematic Reference

The *Schematic Editor Reference* includes detailed descriptions for each tool, command and dialog box option available in the Schematic Editor application. A comprehensive Index is included, making it easy to search for specific information, either by topic or by key word.

Schematic Library Editor Reference

The Schematic Library Editor is a separate Windows application used to create and edit and manage the contents of schematic parts libraries. The Library Editor has many of the same features as the main Advanced Schematic application, plus additional features for component editing. The *Schematic Library Editor Reference* provides detailed descriptions for the design objects, menu commands and dialog box options used with Library Editor, in a convenient separate volume.

The Advanced Schematic component model

In Advanced Schematic, component parts are organized into libraries that correspond to manufacturer data books.

Each component name in the library is associated with one or more component parts descriptions that become the representation of the component on the schematic sheet.

Because components can have multiple devices (e.g., the individual gates in TTL logic components), it is extremely convenient to be able to work with each part of a multi-part component individually while laying out the design.

Libraries of component models

While an image of the component part is "placed" into the schematic, the component information is always stored in the library. Component creation and editing are always performed at the library level, not on the sheet. This approach maintains library integrity and allows library changes to be used to globally update components in existing designs.

As parts are placed in sheets, a back-up image of each component is placed in a special cache, attached to each sheet file. This back-up library allows the user to distribute the schematic sheet files without having to supply the complete set of libraries used to create the design. The back-up library also allows the user to generate a permanent project library for the design. The component images stored in the back-up library can be updated by the source libraries, each time the sheet is loaded in the Schematic Editor.

Connectivity

Connectivity refers to a special feature of the Advanced Schematic environment -- the ability of the software to recognize the physical connections between certain electrical design objects inside the sheet and the ability to associate the logical connections that exist between various sheets in a multi-sheet design. Connectivity is also used to "anchor" objects together when wiring the circuit. For example, you can reposition a part in the sheet and any attached wires will drag with the part as it is moved.

Most importantly, connectivity allows the schematic to perform netlisting and electrical rules checks.

Generating reports

Library Editor tools include two report generators which produce ASCII format text files:

Component report process
Library report process

Opening library files

In the Schematic Library Editor you can load any number of library files, using the Multiple Document Interface (MDI). Advanced Schematic will load the following formats: Advanced Schematic (text and binary formats); Protel Schematic (DOS version 3) .SRC (de-compiled) and OrCAD binary or .SRC (de-compiled) files.

When no open document (file) is displayed, the Menu bar displays three options: File, Info and Help. The Open Sheet File dialog box will display the current directory and any library files with the (default) Advanced Schematic file extension of .LIB. You can change the extension to filter the directory contents. For example, typing "*.*" in the File Name box will display all files in the current directory.

Advanced Schematic allows the use of user-defined filenames (to the DOS limit of 8 characters) and (optional) extensions of up to 3 characters. Extension use is unrestricted -- any extension can be used for schematic sheet files.

File Open process
Protel (DOS) library conversion

Save and Save All commands

Save Edit Save updates the open library file, saving in the default (.LIB) binary format.

Save All The Edit Save All command extends this concept, updating all open library files.

All libraries will be saved in the default (.LIB) binary format.

File Save process
File Save All process

Library and component management

In order to manage and use Advanced Schematic libraries efficiently, it is important that users understand the relationship between libraries, the component names used to access libraries and parts, which are the physical representations of components (symbols), placed in schematic sheets.

The Schematic Library Editor is an independent application for creating or modifying component parts and managing and editing libraries. Under Windows, the Schematic Editor and Library can be run simultaneously -- including multiple instances of either application, memory permitting. While the Schematic Editor and Library Editor run independently, special features provide convenient links between the two applications. For example, you can move directly from a part symbol in the sheet -- to editing its component information inside the source library.

Basic library concepts
Management of libraries
Schematic Library Editor
Component Browser panel

Advanced Schematic Libraries

Advanced Schematic libraries consist of component descriptions, represented by the individual part symbols that are placed in schematic sheets. Components can have one or many parts or subparts (e.g., the gates that comprise a multi-part component like a 74LS00 in the TTL library). The term component always refers to its complete library description -- either a specific manufacturer data book entity or a generic device (e.g., resistor, capacitor, diode, led, etc.).

Many components share the same packaging -- they have identical graphical depictions, but exist as individual names in libraries. Perhaps these are identical devices from different manufacturers, or components that share the same packages but vary on some specification, such as a 120ns versus 80ns RAMs. While it is convenient to access these otherwise duplicate parts using either description, it would be wasteful to create and store a separate graphical version of each item.

Advanced Schematic uses the concept of component groups to associate multiple component names with a single description stored in the library. This keeps libraries efficient and manageable. For example, while the TTL library contains nearly 1800 component names, the graphical and data descriptions that represent these components number only about six hundred.

When a component part is placed in a schematic sheet, the displayed version of the part is a representation of the library version only. The actual component exists only in the library. This means that components and their parts are changed or edited only at the library level -- never at the sheet level. Library level changes are globally applied to each instance of a part when a sheet is loaded.

This principle maintains strict data integrity in parts libraries and has been adopted by Advanced Schematic as the preferred model for most engineering environments, particularly where common libraries are shared by multiple users.

To allow for flexible use of library components, a special cache library is maintained for the current project. Any components used in a sheet are appended to the sheet file, each time the sheet is saved. This file includes a "read only" version of the library component for each part in the sheet. The user can protect the backup library, so that it will not be updated from libraries when the sheet is re-loaded. The backup library allows users to exchange or supply schematic sheet files, without having to distribute all of the component libraries used to create the design.

Component management
Component models
Management of libraries
Schematic Library Editor
Component Browser panel

Part information in the sheet file

While the component description is stored in the library, information is added to the sheet, at each instance when a part is placed. For example, the designator (or label) for each placement is part of the sheet, not a library attribute. Other sheet level attributes include information about the components position in the sheet, orientation, color assignments, 8 user-definable text fields, etc.

At the same time that the part is placed in any project sheet, the library component information for the part is added to a cache. As individual sheet files are saved, the cache information is used to create a backup library in the sheet.

Components are grouped in libraries, by the component type or manufacturer, for convenience.

Components

Each component library consists of three general types of data:

Graphical representations of each component part;

The descriptions of component, including component text fields and other attributes;

A separate listing of component names associated with the component attributes and graphics.

Graphical representation of components

Parts are graphical representations of components, component devices, logic and generic devices. Some components are represented by a single part, such a resisters or diodes, etc. Other components are represented by multiple parts for example, TTL logic components that include multiple gates. In Advanced Schematic, you place each of these parts independently.

Depending upon the component, parts can be displayed using standard ANSI schematic notation, the DeMorgan equivalent (for gates) and IEEE standard notation.

Component description options

Part graphics

All Advanced Schematic parts use vector graphics. This means that each graphic element in parts is defined by its coordinates, line width, etc. -- not by a bitmap array. One key advantage of vector graphics is their device independence. This means that vectors can be displayed or printed at the highest available resolution -- resulting in smooth arcs and angles, while also providing a precision environment for complex graphics. Vector graphics also require less memory when displayed and have more compact library records.

Libraries from OrCAD SDT include display bitmap representations and vector information used by OrCAD SDT for printing and plotting. Advanced Schematic uses OrCAD vector information for both display and output when these libraries are used.

Libraries from Protel Schematic 3 are bitmap images and do not include vector descriptions. When loading Protel Schematic sheets, Advanced Schematic will substitute vector versions of any component names that match the names in Advanced Schematic libraries. Other parts, for example user-generated parts, may exist as unique bitmaps only. These components will be converted to simple vector "block-type" parts when sheets are loaded into Advanced Schematic.

Changing objects graphically

IEEE component representations

Many components in the standard Advanced Schematic libraries have IEEE equivalents. Protel IEEE versions are not superficial shapes, but complete component descriptions that reflect the comprehensive depth of this standard.

IEEE standards allow a number of alternative expressions of component attributes, and each manufacturers data book varies in its presentation of IEEE components. The graphical descriptions for IEEE versions are fully user-editable in the Schematic Library Editor, and a rich set of IEEE graphical tools are included. Protel's IEEE components were compiled from a proprietary natural language environment that yields extremely consistent (and therefore useful) representations.

IEEE equivalents can be specified before or after part placement and IEEE versions can be globally displayed to some or all parts placed in the current sheet.

DeMorgan logic

This option can be specified before, during or after part placement and if desired these equivalents can be globally assigned to some or all logic devices placed in the current sheet.

Parts of a component

Along with graphical depictions, libraries store a number of other component attributes. These attributes are all editable at the library level only -- not after parts are placed in schematic sheets.

Text fields

Each library component has 8 user definable text fields. These fields can hold up to 255 characters. Library text fields cannot be edited from placed parts in schematic sheet files, but can be included in custom Bill of Materials reports.

In addition to the 8 library component text fields, 8 additional user-definable fields are available for each part, at the sheet level. These fields can be displayed or hidden (click Hidden Fields in the Change Component dialog box) and are editable in the sheet, with user definable fonts, sizes and colors. These fields can be up to 255 characters long and are available for Bill of Materials reports.

Component description options

Footprint

Four fields are provided for naming PCB footprint patterns for the component. The first field is the default value and will be used in (Protel format) netlist generation. Four fields allow the user to nominate alternate patterns for SMD versions, etc.

Component description options

Description

A description field of up to 255 characters is provided for part value or similar information. This field is displayed in the component browser when placing parts in the Schematic Editor and can be included in custom Bill of Materials reports.

Component description options

Default designator

A default designator prefix can be stored with each library item. When components are placed, designators will automatically include this prefix.

Component description options

Subparts

Multi-device components include graphical descriptions for each subpart. Each can be independently modified in the Schematic Library Editor.

New Part process
Remove Part process
Previous Part process
Next Part process
Show Normal version
Show DeMorgan version

Pins

Component pins are independent electrical objects. Inside the Schematic Library Editor, the user can define pin names and numbers for each component part.

There are no restrictions of pin placement relative to other component graphics -- pins need not be placed in any particular order nor in contact with any other component graphic elements (body, etc.) to be functional.

Place Pins process
Toggle Hidden Pins process

Hidden pins

Any pin can be defined as hidden. Normally, hidden pins are used for component power nets. These pins are deemed to be connected to the named power nets during netlisting, assuming that power nets with matching labels are available on the sheet. Components vary in their identification of power nets and it is common for labels such as VCC, PWR, +5, etc., to be used interchangeably. One way to avoid missing nets is to wire variously named equivalent power ports together somewhere on the schematic sheet, to avoid missing hidden (and therefor overlooked) power nets. When pins are displayed, they are always deemed to be un-connected and it is assumed that the user will connect these pins manually.

The Hide/Display status of all hidden pins on a part can be changed at any time. Any pin can be defined as hidden in the Library Editor. Normally, hidden pins are used for component power nets. When hidden, these pins are deemed to be connected to the named power nets during netlisting, assuming that power nets with matching labels are available on the sheet. Once displayed, pins are deemed to be un-connected until the user connects them manually.

Place Pins process
Toggle Hidden Pins process

See also

Local hidden pins

Parts on sheet

When parts are placed in schematic sheets, a number of additional attributes are available for editing. These attributes are associated with the specific instance of the part in the sheet only -- they are not library attributes. To change any part, double-click on the placed part or use the Edit Change command. Each editable attribute can be globally edited, which changes applied to some, or all of the parts on the sheet.

Place (part) process

Part type

A part type text field is provided for component part names. This field can be up to 255 characters long and is available for Bill of Materials reports.

Component description options

Designator

Part designators can have their prefix default predefined in the Library Editor (see Default designator, above). The user can override this default manually, when placing or editing parts in the sheet. If no initial numeric value is defined for a designator prefix, it will be placed as U?, R?, etc. Once a numeric value is used, designators will automatically increment to the next digit: U1, U2, etc. Complex cases for multi-device components are also supported, yielding: U1:1, U1:2, U2:1, U2:2, etc. Alpha a numeric designation is allowed. The File Annotate command automatically re-numbers all designators in a project.

Part number

A special text field is reserved for part numbers. This field can be up to 255 characters long and is available for Bill of Materials reports.

Component description options

Library name

This field identifies the source library for the component. It is user-editable at the sheet level and can be included in custom Bill of Materials reports.

Component description options

Colors

Part outline, pin color and (in many parts) fill colors are user selectable attributes. If parts are un-filled, changes made to this field will be ignored. Default colors can be assigned to each part at the library editor level, but these assignments can be overridden when editing placed parts. To apply the local (rather than default library) color, click Local Colors in the Change component dialog box.

About component text

Part text fields are created and stored at both the library level (when the part is defined) and at the sheet level when the part is placed. These text fields can be up to 255 characters long.

Be aware that the length of fields used in netlists: designator (label) type (description) and package (footprint or decal) may be limited, may not support (empty) spaces and may be case sensitive.

The designator, type description and other text fields created at placement can be either hidden or displayed and can independently moved/rotated. Displayed text supports any TrueType font, with user-definable size (in points), style and color. Eight text fields are available when creating component parts in the Schematic Library Editor. These fields cannot be displayed in sheet files, but can be included in expanded format Bill of Materials reports.

Component description options

Opening libraries

The Library Add/Remove command (shortcut: L, A) in the Schematic Editor is used to add libraries to the current open list in the Library Browser. There is no fixed limit on the number of libraries that can be opened concurrently.

Opening libraries is a pre-requisite to opening OrCAD files that have not been previously saved in Advanced Schematic format. Because OrCAD files cannot display components unless all libraries used in the file are accessible, opening sheet files without first opening libraries will result in missing components in the sheet.

File Open process
Basic library concepts

Management of libraries

When a component part is placed in a schematic sheet, the displayed version of the part is only a representation of the library version. Components are edited only at the library level.

Restricting component editing to libraries maintains strict data integrity in parts libraries.

Component cache and backup libraries
Archiving the component cache
Updating sheet parts
Basic library concepts

Component cache and backup libraries

To make library access efficient, a special library cache is created as parts are placed in sheets. A single cache is maintained for all opened projects and this cache holds copies of components placed in any sheet in each project.

Each time any sheet in the project is saved, the cache is used to generate a backup library. This file includes a "read only" version of the library component for each part in the sheet. The user can protect the backup library, so that it will not be updated from libraries when the sheet is re-loaded. The backup version allows users to supply schematic sheet files, without having to distribute their complete component libraries.

It is useful to note that the cache holds copies of all the components currently "in use" in the environment. As you open another project, all of its components are copied from the sheet back-up libraries into the cache. If you remove sheets or close projects the components in those projects are not cleared from the cache. The cache is cleared when you exit the Schematic Editor.

In extreme cases, where many sheets are being loaded during a session, it may be possible to fill the cache to the extent where system performance begins to slow. If this happens, save your projects, exit the editor and re-start the application.

Archiving the component cache

A benefit of caching is the ability to archive the cache contents into a single project library. Because the source libraries may change over time, archiving provides a convenient way of being able to work with an accurate version of the project at some future date. The Make Project Library (shortcut: L, M) command generates a project library based on the current sheet back-up library.

Basic library concepts

Updating

The Library Update All Parts command updates all parts in project sheets to be updated from current versions of source component libraries. This command allows changes in current source libraries to be globally applied to each instance of a component in an entire project.

Library Menus

Most of the menu commands in Library Editor match the form and function of the same commands in the main Advanced Schematic application. Some options are specific to Library Editor.

Menu Commands

File menu

NewOpens a new (empty) library file with the default name LIB_1.LIB. Any number of libraries can be opened at once, limited only by available memory;

OpenOpens an existing library file. The normal mask and path specifications can be used to screen for specific library types, by extension;

MergeOpens a second library and adds all components to the current active library file;

CloseCloses the current library; If the library has not been saved (or saved as) the prompt "Save Library before closing?" will be displayed;

SaveSaves the current active library;

Save As Saves a renamed version of the current active library;

Save All Saves all open library files;

PrintPrints the content of the current active Library Editor edit workspace;

Report Component Generates an ASCII text file report for the current component description;

Report Library Generates an ASCII text file report for the current library;

ExitQuits the Library Editor application.

Menu Commands

Edit menu

These commands work identically to the same commands in the main Advanced Schematic application, applying to library rather than sheet files.

Menu Commands

Place menu

The graphical objects that can be placed from this menu (or from the Drawing toolbar) are identical to the same objects in the main Advanced Schematic application. Special Library Editor commands include:

IEEE Symbol Places special IEEE graphical symbols in the current active library window. These symbols generally conform to ANSI/IEEE standard 91-1984, although some minor modifications have been made to symbol proportions to facilitate on-screen display and printing. Symbols can be placed inside or outside the component body (or outline), without restriction;

Pins Places a single component pin in the current active library window. Pins are automatically designated, as placed. Pins can be freely placed and do not have to be in direct contact with the component body (or outline, as in OrCAD SDT) to be functional;

Text Places a single line of text, similar to the Place Annotation command in the main Advanced Schematic application.

Menu Commands

Component menu

The component menu includes a number of special commands for library editing:

New component	Opens a new, empty component window;
vAuto Create	Opens the Auto Create dialog box. Allows user to create complex components, by describing pin parameters;
Remove component	Removes the current component from the current library;
Rename Component	Prompts user to supply a new name for the current component;
Remove Comp Name	Removes a component from the current group;
Add Component Name	Adds a user-specified component name to the current group;
Copy Component	Copies the current component into another library. User is prompted to supply the name of the destination library;
Move Component	Moves the current component into another library. User is prompted to supply the name of the destination library;
New Part	Adds a subpart to the current component;
Remove Part	Deletes a subpart from the current component;
Component	Next Part/Previous Part -- toggles through the subparts for the current component. Next/Previous/First/Last component -- toggles through components in current library;
Show Normal	Toggles to Normal representation of part;
Show DeMorgan	Toggles to DeMorgan equivalent representation of part;
Show IEEE	Toggles to IEEE representation of part;
Description	Opens the Component Text Fields dialog box. User can specify a type Description, Default Designator (e.g., U? will generate incremented designators U1, U2, etc.), Footprint (e.g., decal or pattern) 1-4 (used to specify pattern for netlist), Text Fields 1-8 (user-definable text, up to 255 characters per field) -- can be added to Bill of Materials (expanded format).

Menu Commands

Options Zoom Windows and Help menus

These menus and commands work identically to the main Advanced Schematic application: applying to library rather than sheet files.

Menu Commands

Tool buttons

Tool buttons are shortcuts for launching frequently used menu commands. The Schematic Library Editor includes a main toolbar, across the top of the workspace and floating toolbars, which the user can position anywhere in the workspace. All toolbars can be hidden. Each toolbar-launched process is available from menus, if preferred.

Main toolbar This toolbar include a number of general purpose commands, such as File Open, Print and Zoom-In, Zoom-Out and Zoom-Workspace. Other buttons are provided for toggling the show/hide status of other toolbars and panels.

IEEE tools Buttons on the IEEE toolbar provide direct access to IEEE graphical symbols used in component creation (Schematic Library Editor).

Library tools This tool bar includes a special collection of tools used during component creation (Schematic Library Editor).

Toggle Main Toolbar process
Toggle Library Toolbar process
Toggle IEEE Toolbar process

Library Editor panel

All component editing is performed at the library level. To change an existing component, or to add a new component to an existing library, you must first open that library from the Library Editor.

Opening a library is similar to opening a sheet file in the main Advanced Schematic application. Libraries can be loaded from Advanced Schematic, OrCAD SDT (in de-compiled .SRC format), or Protel Schematic 3.x (.LIB binary format).

The File New command (shortcut: F, N), opens an empty library named LIB_1.LIB. If the File New command is repeated, a second library named LIB_2.LIB will be opened, etc. Each library will open in its own window, with the library name displayed in the Title Bar. If you save a library in Protel (ASCII) text format, the default extension .ASC is used. Loading and saving of libraries is independent of the file extension used. Users are free to use any extension of up to three non-reserved characters.

Using the Library Editor is a straightforward process of loading a current library (or creating a new library file), searching for existing components and editing those components and their parts or creating new components from scratch. This process can be accomplished totally from within the Library Editor. You can place new components directly into any open sheet from the Library Editor.

When a library is loaded, all component names are listed in the Components window. You can view component parts (each device included in the component record) by moving the selection bar through this list. The four buttons, just below this window allow you to scroll through the entire library. The leftmost buttons move up the component list one item at a time; the left center button moves to the top of the component list; the right center button moves to the end of the list; the rightmost button moves down one component.

The Group window, to the right, displays the names of other components in the current library that share the same component description as the highlighted component name, at left. The Add and Delete buttons, just below the Group window are used to add a new component name to the current group or delete a component name from the current group.

When a component is deleted from a group, it is removed from the library.

The Component New Component command (shortcut: ALT C, N) adds a new component name, (COMPONENT_1) to the current library. If the command is repeated, a second component name, COMPONENT_2 will be added, etc.

Library menus

Creating new components

Components are created in the Library Editor workspace similarly to the same way that schematics are created in sheets. The main difference between these two activities is that the tools used vary with either task. For example, library components to not incorporate wires, buses or sheet entries but they do include pins, which are not placed in the sheet editor.

A component description -- the actual record in the library including the graphical representation of one or more component parts -- is independent of the name or group of names assigned to that description. It is the complete description that is the library entity, not the component name field(s).

When you use the Component New (Component) command, a name is added to the list for the current library, irrespective of whether you have created a description for that name. You can save even libraries with names of components that are not linked to any description. When you load these libraries, choosing these names in the component list box will open an empty workspace.

Advanced Schematic components can have any number of pins or parts, restricted only by the memory available to handle the individual item. Library workspace, where components are graphically created is limited to a "B" size sheet. The user can exercise absolute freedom in terms of the placement of pins, vector objects and even graphical objects (including bitmap formats). Pins do not have to be in contact with the component "body" as in other systems in order to function.

Using imported graphic images in components is allowed, however it is important to note that the graphic record is not stored "in" the library -- rather the library entry "points" to the filename for the graphic. If the path to the graphic object record file changes, Advanced Schematic may not be able to display the item.

Component Description fields

The Component Description command opens a dialog box that allows the user to edit additional fields that correspond to the graphical representation of a component's parts. Fields include:

Description This field is provided to assign a name or component type description to the library entry. This description is displayed when you use the Library Browser as an aid in placing component parts in sheets. The description can be up to 255 characters in length and can be included in Bill of Materials reports.

Default Designator This field is provided to assign a default designator prefix (U?, R?, etc.) to the library entry. The designator is numerically incremented as component parts are placed in sheets. The description can be up to 255 characters in length and can be included in Bill of Materials reports.

Text fields (8) These fields are provided for any user-defined text, up to 255 characters in length. These fields can be used for manufacturer part numbers, house part numbers, price or substitution data, specifications, etc. This text is not displayed nor editable from the Schematic Editor but can be included in user-defined Bill of Materials reports.

Footprint fields (4) These fields are provided for assigning up to four package description footprints (or decals) for the component. This data is used for netlist generation. The first field is the default.

Sheet Part This field allows the user to assign component pin names to nets in other hierarchical sheets. The component then operates as though it were a sheet symbol, with the pins acting as sheet entries. To connect to the other sheet, ports with that sheet name must be present in another sheet in the project.

Component Description options

Component Browser panel

The Component Browser is provided as an aid in searching for, and placing component parts into sheets during schematic editing. This panel can be displayed as needed or hidden, to free additional workspace.

Library Add/Remove
Adding libraries to the current list

Library Add/Remove

The Library List command is used to add or remove component libraries from a current internal list maintained by Advanced Schematic. Holding this list in memory speeds library access, when placing parts in schematic sheets.

When loading OrCAD SDT files, the required libraries must first be loaded into the current library list, prior to loading these files, or components will be missing from the sheets. This is not true for Advanced Schematic files, which have a cached back-up library of components used in the sheet, attached to each file.

Adding libraries to the current list

To add a library to the current library list, choose Library List (shortcut L, L) or click the Library List tool button on the main toolbar.

The Library List dialog box opens. Features of this dialog box include:

Current File List This window lists all currently loaded libraries and is updated as libraries and added or removed from the list.

File Name This window lists the name of the highlighted library from the Files window, described below. You can type the desired library name, if known, directly into this window. The current path and directory are also shown.

Files This window lists files in the current directory that match the current file mask used in the File Name window. You can use this feature to specify the file type, by extension. Advanced Schematic does not restrict the use of extensions to identify library types. OrCAD SDT 3/4 libraries (in decompiled .SRC format), Advanced Schematic and Protel Schematic 3.x libraries can be loaded.

Directories This window changes the current path and directory as you search for the desired library.

To add a library to the current list, move the selection bar through the files listed in the Files window and click the Add button. The library will be added to the Current File List window.

To remove a library, move the selection bar through the Current File List and highlight the desired library, then click Remove.

Click OK to close the Library List window and re-set the updated library list.

Configuration

Managing the configuration of your Advanced Schematic installation is covered under the following topics:

About Preferences
Workspace options

About Preferences

The Schematic Library Editor stores many user settings, such as printer/plotter setups, display colors, and many other options in a special file called LIBEDIT.INI that is automatically added to the Windows directory, the first time that you Exit the application. Thereafter, LIBEDIT.INI is updated each time you exit the application. When you start Advanced Schematic, the program looks for this file and your preferred settings are restored. Other settings belong to a specific workfile, such as the current sheet and color assignments used when the schematic file was created. When you re-load any schematic file, these settings are restored. If you select File New, the normal system defaults or (where applicable) user preferences are restored.

Managing Preferences

You can restore all original (system) default settings by deleting the LIBEDIT.INI file from the Windows directory. You can keep more than one .INI file (with different settings for different types of projects) by temporarily re-naming the file, or by moving it into another directory. For example, you might wish to create a directory that includes all documents for a project with a special .INI file.

Workspace options

Workspace options

The Options Preferences command is used to set-up the workspace design environment.

Workspace colors

Colors can be assigned to workspace display elements.

The Options Preferences dialog box is used to assign colors to selections and the visible grid. These assignments are saved as system defaults in the LIBEDIT.INI files.

Workspace options

Library Editor workspace

The Library Editor environment is very similar to the Schematic Editor. However, the Library Editor does not use standard or user defined sheet sizes. A standard library workspace area is provided. This is equal to a "B" size sheet and can be oriented in either landscape or portrait orientation. You can print the current contents of the Library workspace(s), using the File Print command.

Workspace options

Coordinate system

To provide an absolute spatial reference as you work in the Schematic Editor and Library Editor, a coordinate system is linked to the workspace cursor and to the reference point of each placed object.

Coordinates are displayed at the left end of the status bar and in each object's Change dialog box. Some object Change dialogs list coordinates for individual vector or control points.

Each coordinate unit is equal to .01 inch (.254 mm). The absolute 0,0 coordinate or workspace origin is at the extreme lower left corner of the sheet (or Library Editor workspace). The range of coordinate units is 0-6500.

OrCAD SDT and Protel (DOS) Schematic origins are located at the top left corner of the workspace. Objects coordinate from these systems are converted in Advanced Schematic units, when the files are opened.

Toggle Snap Grid process
Toggle Visible Grid process

Workspace navigation

This section describes how processes and process launchers allow easy navigation of the Library Editor workspace.

Search For String
Jump to Origin

Search For String

The Edit Search For command allows you to conveniently locate a specific text string without having to zoom, pan or scroll through multiple screens

String Type the target string in the Jump to String dialog box.

The cursor will jump to the named string. This option works with free text strings only -- not component text strings. The system will perform three searches:

First -- for a string that matches the specified string in both case, characters and length;

Then -- for a string with same characters and length but ignoring case;

Finally -- for a string with a partial character match (and ignoring case).

For example, typing "Component" would find the string "Component" first. If no match is found it would next find the string "CompONENT" and finally "CompONENTs." When the string is found, the cursor will be relocated to the specified string.

With all of these options, Advanced Schematic will only re-draw the screen if the search target is off the present screen. When a re-draw is needed, the target will be centered in the active window.

Jump to Origin

Jump to Origin Jumps to the absolute (0,0) coordinate. In the Advanced Schematic system, this is the lower-left corner of the workspace (see "Setting a new origin," below).

Changing your view of the workspace

As you click to move from document window (workspace) to window, you are moving the focus of the Windows environment. This also happens when you click in another application window to make that application active. The current focus determines the way that Advanced Schematic tools and features affect your design. One example is the way the current window is affected by scroll bars, panning and zooming.

Scrolling
Auto Pan
Zoom commands
Interrupting screen re-draws
Special navigation keys

Scrolling

Note the shaded bars along the right and bottom sides of the sheet window. Mouse users can use scroll bars to move horizontally or vertically in document windows. The position of the scroll box in the scroll bar indicates the relative position of the displayed portion of the document inside window. Drag the buttons to scroll in real time. Click over or under the button in the shaded area to scroll the visible portion of the screen with each click. Finally, click the arrow buttons at the end of the scroll bar to scroll one unit at a time. You can hide or display toolbars by using the Options Scroll Bars command.

Whenever the document window is sized to display the entire sheet (or library workspace), the scroll bars are automatically hidden, to provide a wider viewing area.

Auto Pan

When the Options Preferences Auto Pan feature is enabled, your view of the current window will automatically pan up-down-left-right as you place, move or draw objects in the sheet. Panning takes place when the cursor reaches the edge of the display when placing, moving or re-sizing objects.

About Preferences

Zoom commands

The Zoom menu commands (and associated shortcuts) also affect your view of the current sheet window. You can set the zoom to view the entire sheet (up to 65 inches square) or zoom all the way in, to about a 4X magnification.

The Zoom command takes advantage of an 8087 (or equivalent) numeric co-processor, if present in your system. In some modes, zooming will be faster if you have an add-in co-processor or a 486DX system.

Three zoom command shortcuts are provided: Pressing pgup increases the screen magnification to the next highest level. Pressing pgdn will decrease the magnification to the next lowest level. Pressing SHIFT+PGUP will increase magnification in 0.1 increments; SHIFT+PGDN will decrease magnification the same amount. SHIFT+PGUP will zoom to the highest magnification; ctrl+pgdn is equivalent to Zoom All.

Menu Commands

Interrupting screen re-draws

Whenever you change the size and/or position of your view of the screen, the contents of the workspace will be re-drawn to reflect the change. You can terminate the re-drawing process by pressing SPACEBAR anytime when the re-draw is in progress. This saves time whenever you wish to immediately scroll or zoom again, without waiting for the re-draw to complete.

Special navigation keys

PGUP	Equivalent to the Zoom In command. Press SHIFT to Zoom at smaller intervals.
PGDN	Equivalent to the Zoom Out command. Press SHIFT to Zoom at smaller intervals.
HOME	Re-centers the screen at the current cursor position.
END	Re-draws the screen at the current cursor position.
Arrow keys	Control the cursor, alternative to using the mouse. Once click moves the cursor one snap grid unit (default is .10 inch).
SHIFT + arrows	Move cursor at 10X snap grid units per click.

Cursor control

The cursor is the special pointing object that allows you to navigate Windows and Windows applications. When running Advanced Schematic, two basic cursor types are provided:

Windows cursor
Workspace cursor

Windows cursor

The familiar "pointer" cursor is used for all standard Windows operations, such as choosing from menus and dialog boxes. This cursor shape is automatically selected when the edge of the document window (workspace) is crossed, as when re-sizing the window, making menu, dialog box or toolbar selections.

Workspace cursor

Inside the workspace, the pointer (Windows) cursor changes to perform object placements and editing. This cursor operates on a gridded system, snapping from grid point to grid point, when Options Snap grid is toggled "on." The workspace cursor will cause the display to pan left/right/up/down when the edge of the workspace is contacted -- as when moving or re-sizing an object. Auto Pan can be enabled/disabled in the Options Preferences dialog box. This dialog box also provides three workspace cursor options:

Large 90 This is a "bombsight" style cursor, useful for aligning placed or drawn object with other items in the sheet;

Small 90 This is the default shape, a small cross with 90 degree orientation;

Small 45 This shape is similar to the Small 90 option, but is oriented at 45 degrees.

Grids

Advanced schematic provides reference coordinate units of .01 inch. These X and Y coordinates are displayed at the left end of the Status line. As you move the cursor inside the sheet, the coordinate display updates constantly to show your position. As an aid in accurately sizing and positioning objects in sheets, two grid systems are provided:

Snap grid
Visible grid

Snap grid

The snap grid defines an array of points in the workspace that restrict cursor movement and the placement of electrical design objects. When using the mouse to control the cursor, you will notice that the cursor moves freely between snap grid points unless you are using the mouse to place or move a selection. When the cursor keys are used, the cursor always "snaps" to the grid. The default Advanced Schematic grid is 10 units (1/10 inch or 2.54 mm). Users can specify a grid from 1 (1/100 inch or .254 mm) to 100 units (1 inch or 25.4 mm).

If you are using the cursor keys to move the cursor, you will find that Choosing SHIFT and a cursor key makes the cursor jump move 10 snap grid points for each click.

Toggle Snap Grid process

Visible grid

A visible grid is provided as a visual reference for placing and moving items. The grid can be toggled between display and hide.

The default visible grid is 10 units (1/10 inch or 2.54 mm). Users can specify a grid from 1 (1/100 inch or .254 mm) to 100 units (1 inch or 25.4 mm).

Options include (default) line grid and dot visible grid. These grids are displayed using the Grid color assignment in the top of the dialog box.

Toggle Visible Grid process

Processes

The Advanced Schematic environment, whether creating and editing sheets, or working in the Library Editor, consists of two basic types of elements: *objects* (or primitives) that describe the data in a design and *processes*, which are used by the system or the user to create, modify, save and report on the data objects. Objects include the sheet workspace, components, pins, wires, lines, graphical images, etc. Processes include menu commands and other discrete processes that manipulate data, such as report generators.

Various methods of launching these processes are provided. For a detailed description of each menu command or other process, go to **Processes** and click on the process name.

Process launchers

Process launchers

A process launcher is any method that the user can access to invoke a process. Various methods are provided:

Menus
Tool buttons
Dialog box options
Mouse and keyboard
Toolbars
Hot keys

Menus and commands

Menus provide access to all commands. Advanced Schematic menu commands are organized to be as consistent with the Windows model as possible. This means that standardized operations, such as opening and saving files, printing or using standardized Windows editing operations such as Cut or Paste is handled in Advanced Schematic using the same methods that other Windows applications use. This makes an application more productive in an integrated environment where the typical user is working with a number of Windows applications. Some processes exist outside the menu command structure. These processes include items like poping-up menus or placing a component from the Browser dialog box. All processes can be assigned to key stroke combinations, using the Hot Keys dialog box.

Menu bar

Mouse and keyboard

Many processes are mapped to mouse and/or keyboard sequences. This provides convenient shortcuts for frequently used operations. A Hot key editing system allows the user to customize hot key assignments. Many generic processes (beyond menu commands) can be assigned. A number of default assignments are provided with Advanced Schematic.

For example, pressing P, J allows you to place a junction without having to open the Edit menu, then choose the Place and Junction commands. Using the LEFT MOUSE button for ENTER and the RIGHT MOUSE button for ESC will allow you to perform many operations without using the keyboard. Some keyboard commands provide the only practical way of performing an operation when you don't wish to move the mouse in the workspace such as choosing a new zoom level while moving a selection.

If you double-click on any placed item, the Change dialog box for that item will be opened, allowing you to edit its attributes.

See also

Hot keys

Menu bar

The Menu bar displays the main menu commands for Advanced Schematic: File, Edit, Place, Library, Current, Options, Zoom, Info, Window and Help. Each menu command and dialog box option is documented in the Schematic Editor Reference.

Short menus
Pop-up Menus

Short menus

When no files are opened, a short menu bar is available, displaying the File, Library, Info and Help menus.

Pop-up Menus

Advanced Schematic includes special default hot key assignments for accessing menu commands. For example, pressing E (when the focus is on the application window) will "pop up" the Edit menu. This also provides a convenient way to avoid having to press ALT to enter a command key sequence.

Processes are available to launch these and can be hooked to hot keys.

Toolbar

Many Library menu commands (and other processes) can be accessed from toolbars. Many of these tool button assignments are shortcuts for Place menu commands.

Place commands are used to create and place an object in a schematic and to create library components, which are special collections of **objects**. Some objects, e.g., library parts are composed of a number of individual primitives or objects that are grouped together and manipulated as a single entity.

The Advanced Schematic Reference and On-line Help system include detailed documentation for each tool.

Main tool bars
Library tools
IEEE tools

Main tool bar

The main toolbar, displayed across the top of the Library Editor application window, includes a number of general purpose tools. This fixed tool bar cannot be repositioned by the user, however it can be displayed or hidden (Options menu).

Toggle Main Toolbar

Library tools

The Library tools palette includes a special selection of general, electrical and graphical tools that support library-level component editing, including pins, arrays, text fields, etc.

Toggle Library Toolbar

IEEE tools

Included special graphical objects used when creating or editing IEEE library component representations.

All toolbar objects and processes can be accessed from menus, if preferred. This allows the user to hide the toolbars (including the Schematic Editor and Library Editor main toolbars), freeing additional workspace for editing sheets or components.

Toggle IEEE Toolbar

Hot keys

Advanced Schematic allows users to create custom hot key assignments for processes and save these assignments in key shortcut files, which can be saved and loaded for specific jobs.

Key assignments can be assigned to any process, not just the processes that are available from menu commands. Key assignments are made from the Options Hot Keys dialog box. Options include:

Keys

Process

Current Process

Description

Assign

Load

Save

Defaults

Setup Hot Keys process

Objects in the workspace

Your design is created by placing component parts (groups of objects that are stored as individual entities in component libraries) and individual objects (wires, buses, junctions, text, etc.) in the workspace. Not all objects are user created or placed. Some, such as Electrical Rule Check error markers and Special strings are generated by the system.

The number of parts and individual objects in a schematic sheet (or project) is limited by available memory only (including virtual memory, supported by Windows 386 Enhanced mode).

For a detailed description of each data object, go to **Data Primitives** and click on the object name.

Creating objects
Selection
Focusing on objects
Changing Objects

Creating objects

A number of different methods are provided for creating and placing objects. For example, you can place individual part any time using the Place Part command or Part tool button.

Object tutorial -- Placing lines
Polyline behavior
Creating other objects

Object tutorial - Placing lines

Object creation is very straightforward in Advanced Schematic. Once you are familiar with a few basic concepts, you will be able to create the full range of objects available in the Schematic Editor and Library Editor applications.

To place a line in the current workspace:

Zoom in on the sheet (press PGUP two or three times) until you can see the visible grid. Choose the Line command from the Place menu (shortcut: P, L or click the Line tool button);

The prompt "Select Line Start Point" is displayed on the Status line. Note that the Windows, pointer-style cursor changes into a cross hair shape. This is the workspace cursor and is used each time you create or move an object.

Click LEFT MOUSE (or press ENTER) once to define a start point for the line;

The prompt "Place Line" is displayed on the status line.

Drag the line segment in any direction. Click LEFT MOUSE (or press ENTER) to end this first segment of the wire;

Note that the coordinates on the status line change as you move the cursor and the end of the line and that the prompt "Place Line" is still displayed.

Move the cursor to continue with a new line segment, which is extended from the existing segment. Click LEFT MOUSE or press ENTER again to define this segment;

If you make a mistake, you can press BACKSPACE to remove the last line segment. You can also press ESC or RIGHT MOUSE to "cancel" the current segment currently being placed.

Click LEFT MOUSE again to end the segmented line;

Note that "Place Line" is still displayed on the Status line. This allows you to end one series of connected lines and then begin a new series of line segments elsewhere in the workspace without having to choose the Place Line command again.

Place Line process

Polyline behavior

The segmented line is a single polyline object. If you position the pointer cursor anywhere directly over the line and click LEFT MOUSE again the line will change. It will now be highlighted (as a dashed line) and at each point where the line changes direction, a small square handle is displayed. These handles mark each vertex in the polyline object. If you position the cursor directly over the handle and click LEFT MOUSE again, you will be able to drag the vertex to a new position and the attached line segments will stretch. Clicking a second time anywhere on, or inside the boundary of a polyline object allows you to move the object to a new position. Click once again to "release" the focus on the object. You can add additional vertices by focusing on a polyline object, clicking on a line segment and pressing INSERT. To remove vertices, repeat this action, pressing DELETE. You can also rotate polyline objects by pressing SPACEBAR as they are moved.

Every object that has its shape created during placement: wires, buses, non-electrical lines, polygons, bezier curves, rectangles, rounded rectangles, arcs and ellipses all share this common polyline behavior as they are placed and graphically edited.

See also

Line placement mode
Selection and Focus

Line placement mode

Advanced Schematic provides two line placement modes that can be selected by pressing SPACEBAR as you create these items. Options include two orthogonal modes and any angle mode.

Any Angle	Allows wire to be placed at any angle;
90/90 Line	Constrains wire placement to horizontal or vertical orientation;
45/90 Line	Constrains wire placement to 0, 45, 90, 135, 180, 225, 270 or 315 degree orientation;

Creating other objects

Other objects are created and placed similarly to wires, buses and lines. These include objects like rectangles, polygons and arcs, ellipses and bezier curves. Some objects are predefined, such as vias or power ports.

For more information about creating and placing these items, see the Advanced Schematic Reference and Library Editor References. For more information about working with objects after placement, see the sections on Selection and Modifying Objects, later in this chapter.

Selection and Focus

Focus and selection provide two distinct and independent methods for changing objects in the workspace. These two methods distinguish Advanced Schematic from other Windows applications where focus and selection are merged in a single operation. For example, other applications display graphical editing handles on each item in a selection, although graphical editing is limited to one item at a time.

Providing two methods of changing items simplifies the display of selections and provides additional control over processes to be performed on individual items and groups. For example, you can focus on, and graphically edit a series of selected single items, without changing the current selection.

Focusing on a single item
Selection
Moving items
Select and De-select commands
Selection and other Windows commands

Focusing on a single item

When you position the cursor over an item and click LEFT MOUSE it becomes the current focus. Only one item can be in focus at a time. This is similar to the way you can change the focus in Windows by clicking on an open window to make it active. Inside Advanced Schematic sheets, you can tell which object is the current focus, because its graphical editing handles, are displayed and the item is displayed in outline form. To move the focus, click on another item or click in a clear area of the sheet to release the focus.

Graphical editing
Handles and polyline behavior
Add a vertex or control point
Remove a vertex or control point

See also

Selection and Focus

Graphical editing

When an item is the focus, you can graphically edit its characteristics. For example, you can change its size or shape by dragging any of the handles. You can also move the object in focus by dragging the center handle (arrows). You can also use the shortcut CTRL+LEFT MOUSE to move an item. Finally, the item in focus can be deleted by pressing DELETE. Moving and deleting individual items is described in more detail under Moving items, below.

Change (Graphically) process

Handles and polyline behavior

Some design objects have special graphical editing characteristics that derive from their polyline behavior. For example, when you place wires, buses or graphical lines, you define a vertex each time the wire, bus or line changes directions. These vertices are displayed as handles when the item is in focus. These items can have complex shapes (hence "polyline"), but can be manipulated (moved, cut, copied, pasted, cleared or deleted) as a single item.

Similarly, polygons have movable vertices and bezier curves have control points which function similarly to vertices.

One key characteristic that all of these objects have in common is the ability to add or delete vertices (or in the case of beziers, control points) from a placed item.

Change (Graphically) process

Add a vertex or control point

To add a vertex or control point to a polyline object:

Click on a polyline object to place it in focus;

Click on the outline (or anywhere inside a solid object). You can tell that the object is in focus, because it will be displayed as an outline, with the graphical editing handles visible.

Move the cursor over any line segment. Click on the segment and press INSERT;

A handle, representing the new vertex (or control point) will be displayed under the cursor.

Drag the handle to define the shape of the polyline object.

Change (Graphically) process

Remove a vertex or control point

Sometimes you will want to remove a vertex (or control point) handle when reshaping a polyline object. To remove a vertex to control point:

Click on a polyline object to place it in focus;

Click on the outline (or anywhere inside a solid object). You can tell that the object is in focus, because it will be displayed as an outline, with the graphical editing handles visible.

Move the cursor over any handle. Click LEFT MOUSE to grab the handle;

The handle will be free to move, under the cursor.

Press DELETE to remove the vertex (or control point).

Change (Graphically) process

Selection

In Advanced Schematic, individual items can be edited one at a time or you can designate groups of items to be changed together.

Selection is performed on both individual items and groups of items and is generally used with clipboard operations: copy, cut, paste and clear. Selection also works with Advanced Schematic's **global editing** feature, which can apply global edits to selected or unselected items only.

Unlike **focus**, selection does not display an object's graphical editing handles. Items are selected, added to a current group of selected items or removed from the current selection using a variety of methods. The simplest method is to move the cursor over an individual and press SHIFT+LEFT MOUSE. This adds the item to the current selection. Pressing SHIFT+LEFT MOUSE again will remove the item from the selection.

Items can be added to (or removed from) the current selection either singly or in groups, using both menu commands and mouse/keystroke shortcuts. Once selected, items can be moved, grouped, un-grouped, exported to another file, cut, copied, pasted into another window or location in the current sheet window or cleared.

Selection and highlighting

When you select an object, its outline color changes from its normal layer color to the Selection color (default black) assigned from the Options Preferences dialog box. The item remains selected until you remove it from the selection using either the De-select command or the SHIFT+LEFT MOUSE shortcut.

Highlighting is related to selection but works within an operation, such as re-routing a wire or when generating a netlist. You can see wires highlight during both of these processes. As objects highlight, you will notice that highlighted item is not displayed in the Selections color but is temporarily outlined in black.

Both selection and highlighting are based on the geometry of objects in the workspace. In other words, physically connected items are included in the selection. This allows you to use selection to trace the connectivity of your design -- like a "continuity" check as you manually route connections and to perform other selection-based operations, such as Cut, Copy or Paste.

Selection is very powerful, but can cause unexpected results, because items remain selected until de-selected. Use the Edit De-Select All command (shortcut: x, a) to clear the current selection before you begin a "new"process that uses selection. If you forget to de-select, use the Edit Undo command to step back to the point of the unintentional result (see below).

Making selections

Selections are made using one of three methods:

Use direct selection: SHIFT+LEFT MOUSE to add or delete individual items to the current selection;

Use the Edit Select, De-select and Toggle Selection commands to define the objects in a selection;

Use the Change dialog boxes to change the selection status of a single object or of a group of objects, by using global editing.

Because items remain selected until they are de-selected, you can combine these methods.

A fourth method achieves the effect of selection without changing the selection status of the items involved:

Choose a process, like the Edit Cut command, then select a single target item for that action.

This method, allows you to perform selection-oriented processes on single items, without requiring pre-selection.

Using the mouse and keyboard

Direct selection using the mouse and keyboard

Direct selection is the most flexible way to designate one or more items to be moved, copied, cut and pasted into a new location or deleted. To select one item at a time:

Hold down SHIFT and click LEFT MOUSE with the cursor position over an item;

The item will be re-drawn in the Selection color (see the Options Preferences command). You can do this repeatedly, each time adding another item to the current selection.

If you hear a "beep" or nothing appears to be selected, try zooming in closer (press PGUP) and make sure that the cursor is directly over the item you wish to select.

The Snap to Center option (Options Preferences command) will cause the cursor to "snap" to the nearest object, when making the selection. This can make it easier to select items from a dense layout, or when zoomed out from the drawing.

To add another item to the current selection:

Hold down SHIFT and click LEFT MOUSE over another item;

To release individual items from the selection:

Hold down SHIFT and click on any selected item.

When released, the item will be re-drawn in its original (layer) color. Other selected items remain selected until they are either individually released (SHIFT+LEFT MOUSE) or until an Edit De-select command is executed.

You can use the Zoom commands (and hot key shortcuts PGUP and PGDN) at any time when making or releasing selections.

Moving items

Items can be moved individually whenever they are the current *focus* -- that is, whenever you position the cursor over an item and click LEFT MOUSE to display an object's graphical editing handles. Click again and the object will be free to move. A final click will place the object at the cursor position.

Moving or deleting one object
Moving selections
Rotating selections
What gets moved?
Bring to front, Send to back

Moving or deleting one object

Note: the special Move Handle, illustrated in the User Guide and Reference manuals is no longer displayed when objects are in focus.

To move a single object:

Move the cursor inside the object and click LEFT MOUSE once to focus the item and display its handles;

Move the cursor anywhere inside the object and click LEFT MOUSE again;

Drag the highlighted object to a new location;

Note that the status line displays the reference coordinates of the object during the move. Items will "snap" to the snap grid during the move. You can rotate or flip most objects during this kind of move (see also, Move Selection, below). To rotate an object 90 degrees press SPACEBAR. To flip the object along its vertical axis, press X. To flip the object along its horizontal axis, press Y.

Click LEFT MOUSE to complete the move. To clear the object from the sheet, simply press DELETE while the item is in focus.

You can click LEFT MOUSE on an empty area of the sheet to remove the focus from the object.

Moving selections

The Edit Move Selection command (shortcut: ALT m, s) allows you to move a complex selection as a single entity.

Moving selections within a routed schematic can sometimes produce unexpected results. For example, if a part is partially within a selection and some wires connected to it are inside the selection, then the wires will be dragged off the part during the move, possibly undoing the connective integrity of the drawing. Use the Undo and Redo commands to recover from an unitended change.

A complex selection containing many items can be moved as a single entity. When you select a Move command the current cursor location becomes the reference point for moving. This process is performed using the Edit Move options:

Choose Edit Move Selection (shortcut: E, M, S);

You will be prompted "Select Reference Point"

Position the cursor at a convenient referen ce point and press ENTER or LEFT MOUSE;

The reference point will be used to provide a convenient way of aligning the selection when completing the move. The selection will be highlighted and free to move. Note how the cursor maintains its relative position to the selection

Move the selection to a new location and press ENTER or LEFT MOUSE;

You will again be prompted "Select Reference Point."

Select another reference point to move the selection to another location or press ESC or RIGHT MOUSE to leave this command.

Rotating selections

While moving a selection, you can rotate it around the cursor and flip it along its x or y axis using the following hot key shortcuts:

SPACEBAR each press rotates the selection 90 degrees clockwise.

X flips the selection along its X axis;

Y flips the selection along its Y axis.

Note: Many individual items need not be selected in order to be rotated. Focus on a single object (component, symbol, etc.) and click a second time to move the object. The commands listed above are effective for most objects.

What gets moved?

If one end of a wire is inside the selection rectangle, only that end will be moved. If the center of pins and junctions fall within the selection, the whole item will be selected with the selection. Wires or parts are only moved if they lie entirely within the selection. If a part with wires is partially within a selection and some wires connected to it are completely within the selection, then the wires will be dragged off the part damaging the connective integrity of the schematic layout.

Bring to front, Send to back

Graphical objects can be placed in sheets so that they overlap. When you place a new item, it is placed at the "front" of other items, by default. When you move items, they retain their position in the display, relative to other overlapping items. Four special move commands are provided for changing the "stacking" order of items in the display:

Bring to Front
Send to Back
Bring to Front Of
Send to Back Of

Select and De-select commands

The Edit Select commands allows you to select all items inside or outside of an area, all items in one sheet, or all objects. You can also select by net or physical connection. These options allow you to extend the selection (or de-selection) beyond a few items.

Edit De-Select provides the same options, less the Net and Connection commands. These commands are used to define complex selections that can be moved, copied or deleted, etc. as a single item. The Edit Toggle Selection command allows you to turn the selection state of individual objects "off" or "on" which duplicates "direct" selection performed using SHIFT+LEFT MOUSE, described above. Shortcut: press X to choose from the De-Select options.

Select and De-select commands include:

Inside Area
Outside Area
All

Importing selections from another schematic

Any part of a schematic sheet can be imported into another schematic file. This is done by:

Opening a new file document window (File Open Sheet command);

Using the Edit Copy command to copy the file (or a selection of the file) into the Protel clipboard;

Using the Edit Paste command to add the file (or selection) to the current schematic.

The imported selection will be displayed with the cursor positioned at the pre-defined reference point. The selection can be moved, rotated or flipped during placement. The following section, "Moving a selection," describes the process of positioning a selection in the workspace.

When a selection has been added to the current layout, any duplication of existing part designators (U1, R1, etc.) is left unresolved. The File Annotate command can be used to automatically resolve all duplicate designators in the sheet.

Selection and other Windows commands

As illustrated above, standard Windows commands, such as Cut and Paste, can be used to manipulate a selection. These commands work with the internal Advanced Schematic clipboard that operates like the standard Windows clipboard.

For example, the Edit Cut command (shortcut: SHIFT+DELETE) will copy a selection to the clipboard and remove the selection from the workspace, and the Clear command simply deletes the current selection and no copy of it is retained in the clipboard.

Under Windows each application has its own range of supported clipboard formats. The internal Protel clipboard is not the same as the clipboard used by other Windows applications. This is because the Protel clipboard stores object attributes, such as net assignment, that are not supported in other Windows applications. However, you can generally Copy or Cut text from another application and Paste that text into Text frame (text) windows.

Using the clipboard - Cut/Copy/Paste

Cutting a selection

The Edit Cut command clears the current selection from the workspace and copies it to the Advanced Schematic clipboard. The Edit Paste command can be used to place the selection back into any open Advanced Schematic sheet window.

Advanced Schematic uses its own internal clipboard format for graphical selections, not the standard Windows clipboard.

To cut the current selection from the active window:

Make sure that the current selection includes only those items you wish to cut;

Use the shortcut SHIFT+RIGHT MOUSE to add items to the current selection or to de-select any selected items.

Choose Edit Cut (shortcut: SHIFT+DELETE);

You will be prompted "Select Reference Point." A reference point is a coordinate relative to the selected item(s). When you paste the selection, the reference point will locate the cursor at this same relative position, allowing you to accurately position the selection.

The clipboard holds the last selection only, each time you use the Cut command, you overwrite the previous selection.

Position the cursor at the desired reference point and click LEFT MOUSE or press ENTER.

The selection will be cleared from the display and copied to the clipboard

When using a mouse, the cursor is not "tied" to the current snap grid. However, when you designate a reference point during Cut or Copy, the grid point nearest to the cursor will be used.

Cut process

Clipboard Reference option

The Options Preferences dialog box includes a Clipboard Reference option which enables/disables the (default) process that prompts the user to designate a reference point when using the Edit Cut and Edit Copy commands.

The reference point pre-sets a cursor position for placing/moving the clipboard contents back into a sheet.

Copying a selection

The Edit Copy command copies the current selection to the (internal) Advanced Schematic clipboard (not the standard Windows clipboard). The Edit Paste command can be used to place a copy of the selection back into any open Protel document window.

The clipboard holds the last selection only, each time you use the Copy command, you overwrite the previous selection.

To copy the current selection from the active window:

Make sure that the current selection includes only those items you wish to copy;

Use the shortcut SHIFT+LEFT MOUSE to add items to the current selection or to de-select any selected items.

Choose Edit Copy (shortcut: CTRL+INSERT);

You will be prompted "Select Reference Point." A reference point is a coordinate relative to the selected item(s). When you paste the selection, the reference point will locate the cursor at this same relative position, allowing you to accurately position the selection.

Position the cursor at the desired reference point and click LEFT MOUSE or press ENTER.

The selection will be copied to the clipboard.

When using a mouse, the cursor is not "tied" to the current snap grid. However, when you designate a reference point during Cut or Copy, the grid point nearest to the cursor will be used.

Copy process

Pasting a selection

The Edit Paste command can be used to place the current clipboard contents into any open Protel document window. Advanced Schematic has its own clipboard format. The standard Windows clipboard is not used.

To copy the current selection from the clipboard:

Choose Edit Paste (shortcut: SHIFT+INSERT);

You will be prompted "Select Location to Place Selection" and a highlighted outline of the selection will be displayed. The cursor position relative the selection is determined by the Reference Point designated when Cut or Copy was used to add the selection to the clipboard

Position the selection in the workspace and click LEFT MOUSE or press ENTER.

You can repeat the Paste command to duplicate the selection.

When a selection has been added to the current layout, any parts that have been added will be not be re-designated to avoid duplication of existing designators. Resolving duplicate designators is done using the File Annotate command.

Paste process

Clearing a selection

The Edit Clear command deletes the current selection from the workspace without copying it to the clipboard.

To clear the current selection from the active window:

Make sure that the current selection includes only those items you wish to clear;

Use the shortcut SHIFT+LEFT MOUSE to add items to the current selection or to de-select any selected items.

Choose Edit Clear (shortcut: CTRL+DELETE).

The selection will be cleared from the display. Use the Edit Undo command (shortcut: ALT+BACKSPACE) to restore the cleared selection.

Clear process

Delete

Using the Edit Delete command is similar in many ways to using selection and the Cut or Clear command described in the previous section. However, with Delete, you simply the cursor over the item, click LEFT MOUSE and it is deleted from the sheet. If you "miss," the system will simply "beep" allowing you to try again. This provides an efficient way to clear several objects from the layout.

Unlike the Edit Clear command, Delete is independent of selection. In other words, the current selection is not discarded when you use the Delete command.

If the Question Delete option is enabled (Options Preferences dialog box), the warning message "Confirm Delete will be displayed as you select each deletion.

All deletions can be restored by using the Edit Undo command (shortcut: ALT+BACKSPACE). If you have deleted a series of items, they will be restored one-at-a-time starting with the last deleted item. The Edit Redo command uses the same first-in/last-out logic. Redo reverses the Undo operations, one-at-a-time.

Delete process

See also

Delete key

Place Array

When you use the Edit Cut (or Copy) command, you are placing a copy of the current selection in the clipboard. The Place Array command provides a powerful way to place multiple duplicates of any clipboard selection into the workspace. To use this feature:

Select the item(s) that you wish to repeat place as an array;

Choose Edit Cut if you wish to clear the selection from the workspace prior to placing the array, or choose Edit Copy if you wish to retain the selection in the workspace;

clipboard. This allows you to pre-designate the cursor position when pasting the selection back into the workspace from the clipboard.

Being careful to keep the cursor on the snap grid, press ENTER or LEFT MOUSE to designate a reference point;

The reference point can be at any position relative the selection and will be used in positioning the items in your array.

Choose Place Array (shortcut: p, y)

The Array dialog box options define the array. Array parameters include:

Placement Variables

Item Count — This option sets the number of times the selection is to be placed.

Text increment — This option is used for designators on pins or parts. Setting this to 1 (default) will assign array designators in-series, for example U1, U2, U3 and so on. This feature follows the same rules as the automatic designator incrementing when placing parts or pins.

Repeated items are placed in a linear array, using the spacing values specified under Spacing.

Spacing

These values specify the horizontal (and/or vertical) distance between each item as it is placed. Each unit is equal to .1 inch. To place an array:

Enter the desired values and click Place. Click OK to store the current parameters for later use or Cancel to return the parameters to their previous settings;

"Select Starting Point For Array."

Position the cursor where you want the array to begin, then press ENTER or LEFT MOUSE;

Use the arrow keys (rather than the mouse) to keep the starting point (and thereby the selection) on the snap grid.

The selection will be placed as an array in the drawing window. If a portion of the array will be placed outside the workspace boundaries the warning message "Selection Is Outside Drawing Window"

You can disable the requirement to provide a reference point when placing a selection into the clipboard (Cut or Copy command) by turning the Clipboard Reference option "off" in the Options Preferences dialog box.

Setup Array process
Place Array process

Changing Objects

The Change command is used to modify specific attributes of placed items. Each item or object has its own range of editable attributes. You can change one item or extend changes across your entire design using powerful global editing options. All changes to objects are simply changes to fields of the object.

Moving and object is modifying its location field(s), either graphically or using a dialog.

To change any placed item, move the cursor over the item and double-click LEFT MOUSE. This shortcut will open the Change (item) dialog box for the selected item.

You can also use the Change command from the Edit menu:

Choose Edit Change (shortcut: E, C);

The prompt "Select (item)" will appear on the status line.

Position the cursor over the target item and press ENTER or LEFT MOUSE;

A dialog box opens, displaying the editable attributes for the item. It is now possible to change any or all attributes of the selected item, such as a wire width or color. These options are described in the next section, Global Changes.

The system will not prevent changes that violate specified electrical design rules. An Electrical Rules Check (ERC) should always be performed before a netlist is generated.

Change process
Global changes
Undo/Redo

Global changes

Changes can be made to a single selected item or they can be applied globally across the entire drawing using flexible, powerful global editing options. Virtually every editable attribute can be globally applied. A simple example would be changing the color assigned to a wire. Typically, the designer would want this new applied to the whole net associated with that wire. Perhaps the color change should be made to a whole range of selected wires. These options (and more) are supported by the Change commands described below. The possible applications for global changes are limited only by the imagination of the design engineer.

The large number of global change options may make this feature appear somewhat complex at first. However, the principles of applying global changes are reasonably simple, once understood. When mastered, this feature can be an important productivity tool that can save a great deal of manual editing of a schematic.

Matching attributes for global changes can be assigned by clicking the Options button in any of the Change (item) dialog boxes. When you click Options, the dialog box expands to display the parameters for global matches.

Each Change (item) dialog box may contain different options since every object type has some unique attributes.

The power of these options can contribute to some unanticipated results -- particularly when complex selections are globally edited. It's always safest to De-Select All (shortcut: X, A), then create a fresh selection **before** you globally edit a selection. Remember, the Undo/Redo features can allow you to recover several operations, if required.

Aligning items
Object colors

See also

Editing objects graphically

Aligning items

Globally editable attributes include the location coordinates for each item. Using coordinates in global editing provides a very powerful system for aligning items in the sheet.

Global changes

Editing objects graphically

If you position the pointer cursor anywhere directly over any polyline item and click LEFT MOUSE, the item will change as it is placed in focus. It will now be highlighted (as a dashed line) and at each point where the line changes direction, a small square handle is displayed. These handles mark each vertex in the polyline wire. If you position the cursor directly over the handle and click LEFT MOUSE again, you will be able to drag the vertex to a new position and the attached wire segments will stretch. At the center of the line will be a special handle with four arrows. This is called the move handle. Clicking on this handle allows you to move the entire polyline object without changing its shape.

Every object that has its shape created during placement: wires, buses, non-electrical lines, polygons, bezier curves, rectangles, rounded rectangles, arcs and ellipses all share this common polyline behavior as they are placed and graphically edited. Editable attributes are described in the Advanced Schematic Reference and Library Editor Reference.

Change (graphically) process

Editing objects using dialogs

Change (Single item) process

Object colors

Each object primitive (everything that is placed) in the sheet be assigned individual color attributes. This allows the user to color code wires, classes of parts or individual items. For example, the user might want to display all VCC net objects in red and GND objects in green. Color assignments can be used for printing schematic sheets as well, if the output device supports the use of color by standard Windows device drivers. Examples include multi-pen plotters and color PostScript printers.

Many types items have a separate color assignment for their outline and fill area. Examples include graphical rectangles, block components and text frames. Items with text fields can have colors individually assigned to each text field as well.

Undo/Redo

Advanced Schematic includes a full multi-level Undo and Redo facility. Each procedure is stored in a stack-like arrangement. When the Undo command is called, the last completed process is undone. Choosing the Undo command again will undo the next-to-last process, and so on. Processes are grouped together in Undo for convenience. For example, if you choose Delete Wire and then delete two wires, undoing this operation will return both of the wires. A new set of processes is started every time you select a new command.

The Redo facility is also multi-level and will reverse all of the previous "Undos" during the current session. Every time you undo something, the operation is stored in a separate stack. If you then select Redo, the last undo operation that you did will be reversed and then the next-to-last and so on.

Undo (and Redo) store each change you make to your sheet from the time you begin the current editing session. To clear the Undo stack, choose Edit Save As. If you perform many complex operations, you may find that system performance will noticeably slow, as Advanced Schematic maintains an increasingly large Undo stack. Clearing the Undo stack will restore lost performance and will also de-fragment and compact the current sheet file, which will also improve response during hard disk access.

Undo process
Redo process

Dot matrix impact printers

Dot matrix printers create letters and images with arrays of dots. The most common of this type of printer creates dots by driving pins against an ink ribbon and into paper. These types printers can produce output quickly but do not provide very good resolution. Resolution refers to the ability of a printer to reproduce an image accurately, usually measured in dots per inch (dpi). A few dot matrix printers produce resolutions up to about 150dpi. Some dot matrix printers have the ability to print colors, please refer to your printer's documentation for more information. If you need to produce presentation quality output, dot matrix printers are not recommended.

Print process

Plotters

Plotters use one or more pens to draw lines on a page. Plotters are especially useful for producing large-scale drawings for presentation. Some plotting devices have the ability to produce color by assigning a different pen number to colors. Windows 3 is supplied with driver support for color plotting. The resolution of a plotter is limited by point size of its pen(s). For example, the smallest dot a plotter can make is the size of its pen's nib. About .3mm is the smallest practical size for fluid ink. Color plots are usually generated using disposable cartridge pins with fiber tips, which are much coarser than high quality pen points. Detail about communicating with and controlling the output quality of pen plotters is provided in the chapter titled Hardcopy.

Using pen plotters presents several problems and limitations. Most pen plotters are vector devices. The Windows environment is biased toward rasterized output and the more contemporary printing technologies that it supports. The reproduction of fonts on plotters, for example is limited by the page description language that a plotter uses to produce output. Reproduction of colors will also be limited to the colors of your plotters pens and the combination of those colors. Unfortunately, most Windows plotter drivers do a very poor job of handling color plotting. Advanced Schematic has been written to work around the tendency of these drivers attempts to "rasterize" the plotter output. For example, in many applications, Windows drivers attempt to recreate the dithering of non-solid colors found on screen. Under the best conditions, plotters tend to produce uneven tones when printing larger areas of color, this is because a plotter usually overlaps pen strokes to fill an area.

Print process

Ink jet printers

Ink jet or bubble jet printers create images by spraying ink on a page. Ink jet printers produce fairly good resolution (about 240 to 300dpi), but are fairly slow. Edges of images and letters can be blurred and softened when reproduced on an ink jet printer because the ink tends to spread out on the page.

Ink jet printers can produce fonts and images of fairly good quality, because most are well supported by rasterized Windows output. The TrueType fonts used in Advanced Schematic will be shown fairly accurately. However, caution should taken when trying to print a design with fine detail, very fine lines will tend to thicken and smaller type sizes will blur or become illegible. Some models are now being produced with "plotter" size sheet capacity. Excellent results can be obtained with color ink jet printers.

Print process

Laser printers

Laser printers have recently become a very popular printing device for producing presentation quality output. These devices provide high quality rendering of line (solid black) and grayscale artwork. Laser printers can produce resolutions from 300 to 600 dpi.

Many laser printers support the PostScript page description language, which allows highly accurate reproduction of TrueType fonts that are supported by Advanced Schematic. Since laser printers use toner instead of ink, edges of fonts and images tend to be very sharp. Laser printers are an excellent choice for presentation quality output. While still relatively expensive, color laser printers are beginning to be widely available, especially from service bureaus in the desktop publishing field.

Print process

Using color

To assign a new color to workspace elements or objects:

Click inside the color box for the workspace element or object;

Object color boxes are found in the Change dialog box for the object. To open the Change dialog box, double-click on the placed object or choose the Edit Change command, then click to select the item.

Choose from over 200 pre-defined colors by clicking on your choice in the Color dialog box; Click OK to complete the color assignment.

Customizing display colors
More about color assignment

Customizing display colors

To customize one of the more than 200 available colors from the Color Selector dialog box: You can set the Red/Green/Blue values between 0 and 255 units.

As you make color assignments, it is important to make sure that your assignments don't conflict in some way that will obscure vital details when you edit your layout. It is recommended that you start with the sheet and selection assignments, if you wish to change a number of the defaults.

More about color assignment

If your system supports 256 colors, your system color palette on will be set to display as many available colors as possible. Advanced Schematic dynamically reprograms the palette as colors are assigned.

Unless the graphics Card/driver you are using is supplied with a Windows 24 bit driver, Windows will use a system palette that allows multiple applications to share color assignments. Windows will take the first 20 colors in this palette for itself and simulate unavailable colors using dithering (mixing two or more solid colors). All 20 defaults are defined by Microsoft and all drivers are expected to provide color matches as close as possible to these.

When using applications that take advantage of 256 color palettes, use of more than the standard 20 Windows default colors may cause "stealing" of colors from the system or from other applications. For example, if you have a bit-map as your background for Windows that used 256 colors then the quality of the display of the bit-map will deteriorate as you select more colors for Advanced Schematic.

In standard VGA and EGA there are only 16 available colors. Windows "takes" all 16 and color requests from applications are either dithered or matched to the nearest solid color. The application can request either dithered or solid colors.

Using fonts

Advanced Schematic directly supports TrueType fonts, including bold and italic formats and scaling. TrueType fonts and supplied with Windows and are available from third-party suppliers in a wide variety of typefaces. Postscript scaleable fonts, and Windows non-scaleable raster fonts can be used, when they are part of vector image files imported into Advanced Schematic sheets.

Note: Some printing or plotting devices do not have full support for scaling and rotation, in particular they may not support vertical text.

Place Text (Single line) process

Image files

The Place Graphic command (shortcut: P, G) is used to import a graphic image into schematic sheets or the library workspace. Bitmap and vector images can be imported and scaled directly in sheets from a variety of graphics formats.

Images are linked during loading, they are not stored in the library. Only the image file path is stored in the library.

Advanced Schematic supports the import of the following image formats;

BMP All uncompressed Bit map images. Windows device-independent bit map format, introduced with Windows 3.0 and increasingly supported by Windows applications.

PCX Paintbrush format, used in Windows Paintbrush and other paint programs and supported by many desktop publishing and graphics programs. Supported colors include, monochrome, 16 color, 256 color, 24-bit color.

TIFF Tag Image File Format, supported by many desktop publishing programs. Supported compression types, uncompressed, LZW, Packbits, Modified Huffman encoding, CCITT Group 3 1D, CCITT Group 3 2D, CCITT Group 4. Supported colors include, monochrome, 16 color, 256 color, 24-bit color

GIF All non-interlaced Graphic Image files.

EPS Encapsulated postscript files with and without display images. If the EPS file doesn't contain a TIFF or Windows Metafile display image then the filename of the EPS image will be displayed.

WMF Only Windows Metafiles which conform to the Aldus Placeable Metafile Format are supported. Most applications which export or import Metafiles support this format.

Because the image "points" to a source file, this file will need to be accessible, when printing. These files must be supplied with sheet files, if you are sending files off-site for printing.

Place Graphic process

Hardcopy

The Library Editor Uses the same set-ups as the Advanced Schematic application. The Print command will print the current component in the following format:

One column will include all Normal display mode parts associated with the component.

A second column will include DeMorgan equivilents (if relevant)

A third column includes any available IEEE representation.

Descriptions, Footprints and 8 text fields are included below the graphical representations.

The Batch Mode option will print this information for all components in the current library.

The Scale to Page option in the Print Setup dialog box will fit all of this information to the sheet size.

PostScript options
Generating a print or plot
Large format and pen plots

PostScript printing

Many laser printers support the PostScript page description language, which allows highly accurate reproduction of TrueType fonts that are supported by Advanced Schematic. Since laser printers use toner instead of ink, edges of fonts and images tend to be very sharp. Laser printers are an excellent choice for presentation quality output. Color PostScript printings are now widely available, as well.

Windows is supplied with drivers for PostScript-compatible imagesetting equipment. This technology, which produces output at 1000 to 2400 dpi, was developed to generate typeset quality images for the graphic arts and printing industries and is widely available in service bureaus. For information about PostScript imaging, contact a service bureau.

See also

More about PostScript prints

Generating a print or plot

Advanced schematic printing and pen-plotting are handled similarly to other Windows 3.0 applications. Windows manages the printing (or plotting) process and provides a range of raster and PostScript printer drivers and vector plotter drivers. These range from 9 pin dot matrix printers and multi-pen plotters, to high-resolution raster imagesetters.

To print or plot from the active Schematic Editor or Library Editor windows, choose the File Setup Printer command. Options include:

Select printer
Batch mode prints
Color mode
Setting margins
Print command
Setup Printer command

See also

Setting-up pen plotters
Scaling prints and plot
Tiling
Options

Scaling prints and plots

Prints and plots can be scaled to a known factor or automatically scaled to fit within pre-defined page margins.

Scale Type a scale factor from .001 % to 400%.

Fit on Page The check print or plot will be expanded or contracted to fit within your pre-defined margins, on the page size selected up for the target printer. The plot will be shrunk or expanded to use the available space, keeping the correct aspect ratio.

Orientation Set the portrait/landscape mode on the printer to match the sheet orientation.

Print process

See also

Tiling Prints
Setting margins

Tiling Prints

When the size of the sheet or library document to be printed exceeds the print area available on the target device, Advanced Schematic will automatically cut the print into two or more sheets or tiles. A pre-defined overlap maintained, so that no area of the print will be lost at the tile edge. You can preview the result of tiling by pressing the Preview button.

It is often possible to reduce the number of sheets required to tile a print, by changing the printer page orientation and adjusting margins. Experiment while in Preview mode to obtain the best match, before choosing the Print/Plot command.

Note: if you have defined too wide a margin for the sheet and have not selected the Scale for Fit option, the sheet will be printed again, for each overlap - so you could end up with as many as four prints - one for each user-defined margin.

Print process

See also

Scaling prints and plots
Setting margins

Print Options

This button opens the standard Printer Setup dialog box, where options for the target device are available. Depending upon the device, options include: sheet size/orientation, the number of copies of each document, etc.

Printer Options

This Options button from this dialog box provides direct access to the Printer's own setup dialog box, also available from the Windows Control Panel. These options add, remove or configure or set-up communications for the specific output device. The available options vary with the features of the selected device. Windows supports background printing from the Print Manager. The queuing of prints and other options can also be controlled from the Print Manager. See your *Microsoft Windows User's Guide* or printer or plotter documentation for additional information.

If printing or plotting from another computer, plots will need to be generated as files. To do this, open the Control Panel's Printer dialog box and click Configure. Select File from the ports menu. You will be prompted to name the file when generating the plot.

This dialog box is supplied by the printer driver and is not generated by Advanced Schematic.

Once all options have been entered, Click Print to proceed with the print or plot, OK to save all the set-ups or Cancel to leave the Printer Setup dialog box without saving the new parameters. As the print or plot is generated (either directly to the output device, or to a filename) the current page and layer being printed is displayed in a dialog box.

If printing or plotting to a file, you will be prompted to supply an Output File Name.

Print process

More about PostScript prints

Some PostScript printers will "time out" and discard the current data when they don't receive the end of page marker within a specified time. This can cause problems where you seem to be missing pages from your plots. If you experience this problem using a PostScript printer or any other printing device then you should go to the Control Panel, select the printer icon, select the printer and click the Configure button. Change the Transmission Retry to 500 seconds, or some other large number. This will allow the printer sufficient time to catch up before the Print Manager gives up.

Print process

Large format and pen plots

Windows is currently shipped with drivers for a number of large format plotters, particularly the new ink jet and electrostatic plotters.

Support is also provided for a number of traditional pen plotters, including Hewlett Packard HP-GL format models. Many other plotters can be configured to support the HP-GL language. See your plotter documentation for more information about available emulation modes.

Current Windows pen plotting drivers, particularly for older model plotters, have a number of reported limitations. This is because some drivers, including the generic HP-GL driver, treat vector pen plotting similarly to raster plotting. This results in a number of problems, for example, poor rendering of filled arcs, inefficient (slow) plotting and other problems, especially when handling complex geometry. New or improved plotter drivers are reportedly under development for some types of plotters. For up-to-date information about plotter driver support under Windows, contact Microsoft or your plotter manufacturer.

If you plan to do your plotting from another computer, you will need to generate your plot as a file. To do this, open the Control Panel's Printer dialog box and click Configure. Select File from the ports menu. You will be prompted to name the file when you generate the plot.

Print process

Setting-up pen plotters

The options available will depend upon the type and model plotter selected. Guidelines should be documented in your plotter manual. Most plotters will require the following setup decisions:

Pen speed

Determining the correct pen Speed is largely a matter of trial-and-error. Some users may find they have to choose a slower speed to get properly "filled" solid areas. The condition of the pen points, freshness of ink, etc., can have a significant impact on plot quality. Some plotters have force and acceleration options in addition to pen speed. Consult you plotter manual for recommended setting for the paper or film and pen combination you intend to use.

Assigning pens

If generating pen plots for a multi-pen plotter, you can assign different pens to different colors. Pen size and pen number assignments are made from the Printer Options dialog box.

Producing good quality pen plots
Plotter pens and plotting inks
Setting the pen speed
Communications with Serial plotters
Baud rate, data bits, etc
Communication problem
Print process

Producing good quality pen plots

Pen plotters can be used to produce very sophisticated design artwork, when the many variables affecting plot quality are understood and applied to the process. But, there are inherent problems with pen plotting that need consideration. The variables that directly effect plot quality include:

Accuracy of the plotter -- particularly its "repeatability" or ability to return accurately to specific coordinates, over the entire plot area;

Type and condition of plotting pens;

Plotting film or paper;

Type and age of the ink selected;

Environmental factors - i.e. temperature and humidity;

Pen speed and pen size settings.

Other factors include the experience of the operator and the maintenance and storage of equipment and materials. Following a few simple rules ensures that the best quality possible is obtained.

Perhaps the most important factor is the quality of the paper (or drafting film) and the pens that you use. Use inexpensive paper and fibre tip pens for check plots - save the best pens and film for the final plot.

Print process

Plotter pens and plotting inks

There are a wide range of plotting pens on the market. Felt, and plastic tipped pens are convenient to use, but only suitable for draft plots. Pens used for master artwork must be capable of providing a consistent ink flow, must not dry out when the pen is lifted off the film for short periods and must be of the correct diameter for the selected plot scale.

The pens that have been found to be the most suitable are those with tungsten carbide, cross-grooved points. A latex-based ink will provide a dense plot without the ink running or drying out in the pen. Your local plotter supplier will make specific recommendations.

Setting the pen speed

Pen speed is a critical, and often overlooked factor in plot quality. It will be worthwhile to make a series of experimental plots to determine the optimum settings for your combination of plotter and materials. You may also improve the plot result by making small adjustments to the pen size selection. Slight changes will adjust the amount of "overlap" obtained when filling in solid areas -- with further adjustments needed as the pen wears during normal use.

Communications with Serial plotters

Most plotters are controlled via an RS232-C (serial) interface. A cable connects the plotter and computer to provide two-way communication. Correctly configuring this combination of computer software, serial port, cable and plotter can be a challenge, even for experienced engineers.

If you are installing a serial plotter for this first time, this section explains the relevant RS232-C conventions.

The RS232-C standard defines the signals for bi-directional communication where there is no inherit distinction between the computer and the output device. In the jargon of serial communications both devices are referred to as DTE, or Data Terminal Equipment. Signals, such as Transmitted Data are assigned to the same pins in both devices, unlike the parallel standard where each pin has a single function.

Each serial "terminal" needs an intermediary device or devices to connect the "transmitted" data pin of one DTE to the "received" data pin of the other, and vice versa, and to correctly configure the handshaking signals.

These intermediate devices are called Data Communications Equipment (DCE), which connects to DTE, transmits and receives the data over a channel but is neither the source nor the final destination of the data. A modem is a DCE - it both modulates data for transmission over a single voice channel and demodulates it back to digital data.

More about plot communication
Having problems?

Baud rate, data bits, etc

Once a correct serial connection between the computer and plotter is achieved, the correct communications parameters must be selected.

Windows allows you to change these settings using the Control Panel Printer dialog box.

Your plotter manual should indicate the default settings of the plotter and will contain information on changing the communications setup. Some plotters do not have default settings, as such, but use DIP switch settings which must be configured before the plotter is operated.

You will need to match these parameters using the Printer dialog box. Once set, these settings are stored with your Windows preferences (Exit Windows, enabling the Save Settings option).

A baud rate of 2400 bps is standard for many plotters, and a good place to start, if you don't know the specific recommendations for your plotter. This is an intermediate baud rate and should yield error-free data transmission with cables up to 50 feet (15 meters).

Your plotter manual should also document interfacing and handshaking settings.

See also

Having problems?

Pen plot problems

If you are having problems plotting and you are confident that you have the right cabling and parameter settings, check the following:

Inspect the cable connections and make sure that no wires have broken. Also check that your Windows settings match the plotter baud rate, parity, etc.;

Confirm that you are using the selected serial port;

If your plot progresses normally at first, then starts putting stray lines or arcs all over the layout, this generally indicates improper handshaking. You may also have a problem with one or more pin assignments and your cabling may need modification;

Another possible solution is to keep the plotter cable as short as possible and keep it away from power cords and other "noise" sources.

If you are using a long cable, you may have to reduce the baud rate to obtain error-free transmission. Due to the distributed resistance and capacitance of cables, there is a trade-off between cable length and baud rate for reliable data transmission;

Remember, if you change the communications settings at the plotter, you will have to match the new settings in Windows under the printer's Set-up dialog box.

Make sure that you have specified the correct plotter driver. For example, is your plotter a "true" Hewlett-Packard (HP-GL) or a "compatible"? Many plotters emulate HP-GL in addition to their own plotting language. If you are using a dual-language plotter you may have to configure the plotter for the correct language. This is done using Control Panel or dip switch settings, depending upon the plotter type and model. See you plotter manual for details.

Finally, erratic plotter behavior can be the result of plot file corruption -- usually the result of a disk failure or system error during file creation. If you have been unable to solve your plotting problem, try plotting one of the (supplied) demonstration files, as a cross-check.

OrCAD compatibility

Both schematic sheet files and libraries from OrCAD SDT 3/4 can be used with Advanced Schematic. OrCAD SDT 3/4 3 files are loaded using the File Open Sheet command, just like Advanced Schematic files. All OrCAD SDT 3/4 design objects are supported by the Schematic Editor. When you choose File Save, the file will be saved in any of the three following formats: Protel binary (the Advanced Schematic native format), Protel ASCII (a text version of the Advanced Schematic format) or OrCAD binary.

SDT libraries
Basic library concepts

OrCAD libraries

OrCAD SDT 3/4 component libraries can be converted into Advanced Schematic libraries. The first step is to convert the OrCAD format library into its decompiled (.SRC) format. Load the decompiled version into the Protel Schematic Library Editor and then save in Protel Binary Format.

Basic library concepts

Protel Schematic 3 (DOS) compatibility

As with OrCAD SDT 3/4 files and libraries, both schematic sheet (.S**) files and libraries from Protel Schematic 3.x can be used with Advanced Schematic. Protel Schematic 3 files are loaded using the File Open Sheet command, just like Advanced Schematic files. When you choose File Save, the file will be saved in any of the three following formats: Protel binary (the Advanced Schematic native format), Protel ASCII (a text version of the Advanced Schematic format) or OrCAD binary. There is no option for saving files in the Protel Schematic 3 format. Users of earlier version Protel Schematic or Tango Schematic system must first convert files to Protel Schematic 3 format before these files can be opened in Advanced Schematic.

Schematic 3 libraries
Basic library concepts
Protel (DOS) library conversion

Libraries (Schematic 3)

The way that Advanced Schematic handles libraries, components and library editing is fundamentally different than the way these functions are performed in Protel Schematic 3. Please refer to the relevant sections of this guide for detailed explanations regarding Advanced Schematic library concepts.

Protel Schematic libraries are organized into a "flat" structure where each component has a unique description. All graphical representations of Schematic 3 library components (except their pins) are rendered as bitmap images. Advanced Schematic library parts are vector graphic based. When converting files into Advanced Schematic format, the differences between the two library systems must be reconciled to preserve connectivity. This is done in two basic ways: a library of equivalent vector (Advanced Schematic) components is maintained. These components are automatically substituted for like-named standard Schematic 3 versions. For custom components, a substitute vector component is created.

User created bitmaps (Schematic 3)

Protel Schematic 3 libraries and sheets may include user-generated bit map graphics that cannot be substituted from PROTEL.LIB when Schematic 3 sheet files are loaded into Advanced Schematic.

A special library and vectorization support is provided for converting Schematic 3 user libraries into Advanced Schematic format at the time the sheet is loaded.

Basic library concepts
Protel (DOS) library conversion

Pin Editing (Schematic 3)

Schematic 3 allows users to edit pin attributes, including pin positions, on the fly, from inside the schematic editor. This is not possible in Advanced Schematic, where all changes must be made at the library level, using the Schematic Library Editor application. Because the Library Editor can be run at the same time the Schematic Editor is running, it is a simple matter to switch tasks and make library level changes, then return to the editor. Special links are provided that allow the user the move directly from one editor to another. For example, from the Library Browser window in the Schematic Editor, the Edit button will switch the user directly into the Library Editor. The library for the selected part will be opened and the component will be displayed in the edit workspace. Similarly, pressing the Place button on the Library Editor main panel will return the user directly to the Schematic Editor, with the part ready to place in the current sheet window.

Basic library concepts
Protel (DOS) library conversion

ASCII file formats

Advanced Schematic sheet and library files have default binary formats. These binary files are compact and efficient when loading or saving. To make it easier for users to directly access data, the system includes options to generate ASCII versions of these files.

Other files are generated in ASCII format only. These include netlists, Bills of Material and other reports.

Detailed information for ASCII version sheet and library files can be obtained by contacting Protel Technical Support.

Advanced Schematic Glossary

annotation Component reference designators (or labels) that appear on the schematic sheet, in netlists or on a printed circuit board.

ANSI International standard for technical drafting. See also ISO.

any angle Non-orthogonal drawing mode where (non-electrical) lines or wires that can be placed at angles other than 45 or 90 degrees.

arc Circular or semi-circular design elements. Protel for Windows generates arcs of degree resolution.

array Multiple instances of a single item, placed using the Place Array command.

ascend To move from a child sheet, back to its parent sheet in a hierarchical design.

ASCII American Standard Code for Information Interchange. Standard seven bit code for representing alphanumeric data and computer instructions.

attributes The characteristics of an item that can be edited, or changed. For example, wire attributes include width, color, etc.

back annotate Updating schematic information from changes made to the printed circuit board layout.

backup library A special library that includes component records for each part placed in a schematic sheet. This library is attached to the sheet file when the sheet is saved.

bezier curve Complex graphical curves, placed as precision arcs defined by a series of control points.

Bill of Materials Or, BOM. A list of the components (including quantities).

body See outline.

border Graphical device outlining the edge of the sheet workspace, normally including coordinate references for the sheet.

browse Viewing library names from the Protel for Windows workspace.

bus A special wire type that symbolically represents a collection of individual nets.

cache A temporary record of each component used in a project. This cache is used to create a backup library, attached to each sheet file. See also, backup library.

child A subsheet descended from another (parent) sheet in a hierarchical design.

Clear To remove a selection permanently from the workspace. Same effect as Delete. See also Cut, Copy.

clearance Air gap specified for routing a net. Field of the directive primitive

clipboard Reserved memory used to hold Cut or Copy command selections.

clock Pin type that symbolically represents clock function, represented by a small triangular shape at the pin base.

command Any process that is performed by choosing a menu option, e.g., Place Component or File Save.

comment Optional component text field created when a component is placed. Normally used to hold a component value, description or part number.

complex (in hierarchy), when a sheet can appear more than once in a project.

component The specific schematic library package that represents either a manufacturer catalog listing or a generic type. See also part.

component field Any of 8 different library fields that hold optional component text.

component txt Text that is associated with a schematic part or PCB component.

connection The logical or physical link between any two netlist nodes.

connectivity The logical relationship of components and wires.

control point The special graphical handles used to define a bezier curve.

de-select Releasing the selected condition of an item (or group) in the document window.

default Program settings or options that remain activated until changed by the user.

descend To move from a schematic sheet, to a child sheet, represented by a sheet symbol, in a hierarchical design.

designator Also called component label. The unique identifier assigned to each component in a circuit.

device See part.

directive A special symbol used to attach PCB layout information to a specific net, such as routing priority, track width, etc.

discrete Generic component types such as transistors, capacitors, etc.

dot Pin type that symbolically indicates inversion, represented by a small circular shape at the pin base.

DXF Format for AutoCAD files.

electrical rules Connectivity-based features that check for shorts, undriven inputs, unconnected wires and similar electrical design violations.

fill Color or pattern assignment for the inside of graphical items.

flat design A non-hierarchical multi-sheet schematic where Ports are used to indicate connections to other sheets.

fwd annotate Updating a printed circuit board layout with changes made to the schematic.

focus The current active individual which displays its graphical editing handles. The object in focus can be moved, deleted or re-sized. See also selection, handle.

global In relation to the use of net identifiers (net label, port, sheet entry), refers to identifiers whose scope includes an entire project. See also local.

global change Any change that can be assigned to like attributes of other primitives of the same type.

group Refers to one or more library component names associated with the same component description. Library entries can have any number of component names that share the same specification and graphical description.

handle Small arrows displayed on an object that is the current focus.

hardware arc When plotting, arcs which are created by the plotter, from coordinate, line width and radius information.

hidden pin Component power-type pin that is normally hidden when the component is

displayed.

hierarchy The concept of hierarchy applies generally to any multi-sheet schematic. Sheet Symbols are placed within one sheet to represent another sheet (or subsheet).

highlight A special display state that outlines items as an aid to identification or editing. For example, when placing or moving wires, they are displayed in a highlighted condition. See also selection.

hot key Any key that has a process assigned in Advanced Schematic.

IEEE International ANSI/IEEE Standard 9-984 for graphical representation of circuits.

IGES Initial Graphics Exchange Specification. A general use standard for exchanging graphical information between platforms or applications.

intersheet (connection) In multi-sheet flat designs, these are connections that represent a net that crosses from one sheet to another.

ISO International Standards Organization. See also metric.

isolation See clearance.

junction Special symbol, used to electrically (logically) connect two wires.

label See designator.

library Collection of components, devices or symbols, stored in Advanced Schematic library form.

line Any line that is non-electrical in the drawing. Electrical lines are referred to as wires.

link The process of listing all associated sheets for a design.

local In relation to net identifiers, refers to net labels whose scope is limited to a single sheet.

master sheet The "topmost" sheet in a multi-sheet design project.

merge To move components from one library to another.

min X, Y The minimum X or Y coordinate of items in the Protel for Windows workspace. This describes the left-most and bottom-most coordinates used in the file or plot.

Module port See port.

multi Multi-device component. Schematic component whose devices (gates) can be represented by individual logic symbols.

net label Special text symbol used to associate a wire with a specific net.

netlist A text file that lists all the connections of an electronic circuit.

object Any individual item that can be placed in Advanced Schematic sheets.

orthogonal Drawing mode, where wires are constrained to either vertical, horizontal or 45 degree placement. See also any angle.

outline The symbolic shape used to describe individual components or devices.

package The physical description, or "footprint" of a component, e.g., DIP6, defined by the number and location of pins, dimensions, etc.

parent Any sheet that includes sheet symbols for another subsheet (child) in a hierarchical design.

part field Optional text fields added when placing the component. See also component field.

part Graphical representation of a library component or one part of a multi-device component.

pin Graphical representation of a component pin on schematic component symbols. Schematic pins are connected using wires.

pipe-link An OrCAD design element, used to list the subsheets associated with a master sheet in a flat, multi-sheet design.

polygon Multi-sided filled graphical (non-electrical) object.

polyline Lines that can be rendered with multiple joined segments and manipulated as an individual entity. Wires, buses and graphical lines are all polylines in Advanced Schematic.

port A symbol used to indicate a connection to another sheet in a flat design.

power port Special component symbol, normally used to indicate a power source.

primitive Refers to any individual object, or object primitive. See object.

process Any event generated by Advanced Schematic, including menu commands, dialog box options or shortcuts that either changes an attribute of an object, imports data into, creates data or exports data from Advanced Schematic.

process launcher Any user initiated action that begins a process.

project All the sheet files that are associated within a single design, irrespective of their hierarchical or flat organization.

Project Mgr Advanced Schematic system for opening, saving and manipulating multi-sheet schematics.

project window Special windows displayed at the left of the sheet window when the Project Manager is active. This window displays an image representing each open sheet file.

root sheet See master sheet.

selection A method of grouping items which are manipulated as a single entity.

serial Refers to RS-232C and RS-422 standard for data terminal equipment (DTE) communications.

sheet An individual schematic file, displayed in its own window. Each schematic sheet, including hierarchical subsheets, are saved as individual files.

sheet symbol A graphic representation of a schematic sheet that can be placed on another sheet, indicating hierarchy.

signal Any net. Generally used to refer to any non-power net.

Simple (hierarchy) where each sheet in a project is unique. See also Complex hierarchy.

snap grid An invisible array of regularly spaced points on the screen that defines the possible cursor positions.

software arc When plotting, arcs which are generated by Protel for Windows using straight line chord segments. See also hardware arcs.

Status line The window at the bottom-left of the screen that displays the current X and Y cursor coordinates, defaults; and user prompts.

step-and-repeat See array.

stimuli A special symbol used to indicate the point at which the function of a signal (expressed as an algorithm) is applied to a circuit during simulation.

string Individual element of free or component text.

subpart see device.

subsheet In multi-sheet schematics, any schematic sheet file associated with a master or parent sheet. See also hierarchical design.

symbols Various graphical and electrical or logical objects placed on the schematic to represent components, devices or blocks of circuits.

test vector A special symbol used to indicate the point at which a non- algorithmic stream of signal values is applied to a circuit during simulation.

text frame A graphical object placed on the schematic sheet that holds up to 32,000 characters of formattable text. Parts, netlabels and other objects have their own text fields of up to 255 characters.

title block Area of the schematic sheet reserved for the drawing title, revision information, etc.

tool Any menu command that is available as a push button on a Tool palette.

trace see probe.

vertex The joint of any two straight line segments in polyline objects: wires, buses, lines and polygons.

visible grid A user-definable display layer that provide a visual reference for positioning items accurately on-screen.

wire An electrical (logical) conductor in the schematic drawing, represented by a special line type.

worksheet See sheet.

SLM.EXE (Schematic 3)

Advanced Schematic Library Editor Application.

To create, edit and manage your component libraries,

Choose File Open. You can also launch the Library Editor as a separate application from the Advanced Schematic (sheet) Editor by pressing the Run Library Editor button from the Main toolbar or by choosing the Run Library Editor command from the Library menu.

Basic library concepts
Protel (DOS) library conversion

Schematic Library Editor Help

Select Printer

The available output device options will include those that have been installed using the Windows Control Panel (see your *Microsoft Windows User's Guide* for details). Most devices are supported by drivers delivered with your Windows software. You should note that new and updated drivers are released for both new and existing devices on a regular basis. For the latest information about print drivers, contact Microsoft Windows support or the device manufacturer.

Rotation of fonts is not supported for all printers and the substituted fonts will only be used if the text on your schematic is in a standard horizontal (or landscape) orientation, and within the size capability of the printer. PostScript printers support rotation of fonts at any angle

Setup Printer process

Batch Mode prints

When using this option from the Schematic Library Editor, this option prints either a single (current) component or a batch of all components in the current library. This option allows you to print out an entire component library in a single operation. When you choose this option, all representations of a component are printed, including each part (or device) DeMorgan and IEEE equivalents, when applicable. Component description fields are also added to the sheet. This option works with all other print/plot options, including scaling, etc.

Print process

Color mode (printing)

Two choices are available: color mode takes Advanced Schematic screen color assignments and uses these to assign colors to the print or plot, based upon the options available in the print or plot driver. Monochrome PostScript or HP-PCL devices will print grayscale representations of color. The number of gray levels, and the assignment of color to grayscale depends upon the driver and device. The Monochrome option, prints images in solid black/white only. No dithering or grayscale support is provided. This option is appropriate for low resolution dot matrix and single pen plotting.

Print process

Setting margins (printing)

The user has total control over margins, limited only by the margin limits built into printers or plotters that do not allow printing to the sheet edge (e.g., PostScript printers). When used with the Scale and Scale to Fit Page options (described below), this option will size the print area to fit as closely within the margins as allowed by the aspect ratio of the print area. The Preview option allows you to preview the result of all settings and make adjustments before printing.

Print process

See also

Scaling prints and plots
Tiling prints

Basic library concepts

Libraries consist of component descriptions, represented by the individual part symbols that are placed in schematic sheets. Components can have one or many parts or subparts (e.g. gates in TTL components). The term component always refers to a complete library description -- either a specific manufacturer data book entity or a generic device (e.g., resistor, capacitor, diode, LED, etc.).

Many components share the same packaging -- they have identical graphical depictions, but exist as individual names in libraries. These can be equivilent devices from different manufacturers, or components with the same package but a different function (e.g. 120ns versus 80ns RAM). While it is convenient to access these otherwise duplicate parts using either description, it would be wasteful to create and store a separate graphical version of each item.

Advanced Schematic uses the concept of component groups to associate multiple component names with a single description stored in the library. This keeps libraries efficient and manageable. For example, while the TTL library contains

nearly 1800 component names, the graphical and data descriptions that represent these components number only 600 or so.

When a component part is placed in a schematic sheet, the displayed version of the part is a representation of the library version only. The actual component exists only in the library. This means that components and their parts are changed or edited only at the library level -- never at the sheet level. Library level changes are globally applied to each instance of a part when a sheet is loaded.

This principle maintains strict data integrity in parts libraries.

Component management
Component models

Keys (Hotkey)

Select the key (or mouse button) for hot key assignments. Assignments can include the ALT, CTRL and SHIFT keys.

Setup Hot Keys process

Process (Hotkey)

Selects the process for hot key assignment.

Setup Hot Keys process

Current Process (Hotkey)

This box lists the current process assigned to the selected key combination.

Setup Hot Keys process

Description (Hotkey)

This box describes the purpose or use of the current process.

Setup Hot Keys process

Assign (Hotkey)

Adds the key and process selections to the current hot key file (.KEY extension).

Setup Hot Keys process

Load (Hotkey)

Loads an existing key file (.KEY extension) for editing. The Library Editor has its own hot key set, independent of the default Advanced Schematic set, named LIBEDIT.KEY.

Setup Hot Keys process

Save (Hotkey)

Updates the current key file (.KEY extension) with items assigned during the current editing session. Overwrites any previous assignments in the current hot key file.

Setup Hot Keys process

Defaults (Hotkey)

Resets the default hot key assignments. Any new assignments that have not been saved as a new hot key file will be over-written.

Move Handle

The Move Handle, referred to in the reference manuals and *User Guide* is no longer displayed when the object is in focus. You can now move (focused) objects by placing the cursor anywhere inside the boundary of the object and clicking LEFT MOUSE. When the object is in the desired position, click LEFT MOUSE again.

See also

Selection and Focus

Inside Area (Select)

Allows you to define a rectangular selection area. Only those objects that lie completely inside the area are included.

To choose the items inside a selection rectangle:

Choose the Edit Select Inside Area command (shortcut: s, i);

You will be prompted "Select First Corner."

Move the cursor then press ENTER or LEFT MOUSE to define the first corner of the selection rectangle;

The prompt changes to "Select Second Corner."

Move the cursor to enclose the selection area in the highlighted rectangle;

Press ENTER or LEFT MOUSE to complete the selection.

The newly selected objects will be highlighted using the selection color. Any previously selected items will remain selected until de-selected.

See also

Outside Area

Outside Area (Select)

This option selects everything in the sheet outside the selection rectangle. The rules for inclusion in the selection are the same as for the Inside Area command. The procedure for defining the selection rectangle is the same as for **Inside Area**.

All (Select)

This command selects everything placed in the sheet.

Connection (Select)

Choose Select Connection (shortcut: S, C) to select the free objects (wires, buses, junctions,) that connect using the physical geometry of the system. Non-electrical objects will not be selected, even if they touch the connected items.

Toggle Selection

The Edit Toggle Selection command allows you to turn the selection state of individual objects "off" or "on" which duplicates "direct" selection performed using SHIFT+LEFT MOUSE. Shortcut: press X to choose from the De-Select options.

To use this feature:

Choose the Edit Toggle Selection command (shortcut E, N);

You will be prompted "Select object"

Click on an object type to add or remove the item from the current selection;

The prompt "Select object" will be continuously displayed.

Press ESC to leave the Toggle Selection command.

See also

Select and De-Select commands

Bring to Front

This command moves an object to the front of other items in the display. When you use this command, you are prompted to choose the item to be moved. When you click on the item, it moves to the front of the display without changing its x or y coordinates.

Send to back

This command sends an object to the back of other items in the display. When you use this command, you are prompted to choose the item to be moved. When you click on the item, it moves to the back of the display without changing its x or y coordinates.

Bring to Front Of

This command moves an object to the front of another item. When you use this command, you are prompted to choose the item to be moved. When you click on the item, you are then prompted to choose the "target" item. The item to be moved will be re-located in front of the "target" without changing its x or y coordinates.

Send to Back Of

This command moves an object behind another item. When you use this command, you are prompted to choose the item to be moved. When you click on the item, you are then prompted to choose the "target" item. The item to be moved will be re-located behind the "target" without changing its x or y coordinates.

Cursor shape

Options include: large 90 degree cross (cross hair type), small cross or 45 degree cross.

Snap to Center

If Snap to Center is enabled the cursor will jump to the center of a pin, wire, etc., as it is moved. If you have the Snap to Center off, the cursor position will remain relative to the point where the selection was made.

When you select a component with Snap to Center Option on, then the cursor will jump to the reference point of the component (generally pin 1). If this option is off then when you select a component the cursor will remain in the same position relative to the component.

Save Defaults

This option allows user settings, preferences and defaults to be written to SCH.INI and LIBEDIT.INI files when exiting the Schematic Editor and Library Editor applications. These settings, preferences and defaults are then loaded the next time you run Advanced Schematic.

Auto Backup

Sets interval (in minutes) for generating an automatic backup of the active workfile. To disable this feature, type 0. The default interval is 30 minutes.

Auto Backup File Count

Auto Backup works on the basis of a rotating file system. This option allows you to specify the number of backup files that will be automatically created as you work. For example, if you have a file called TEST.SCH then the first backup will be saved as TEST.AB0. At the backup interval, a second backup, TEST.AB1, etc., up to the total number of files selected. The next backup would again be TEST.AB0, overwriting any previous version with that name. The default value is 1.

Dialog boxes

Dialog boxes launch processes and other perform other functions, such as changing the attributes of data primitives. An example would be using a Change dialog box to enter new coordinates for an object. Some dialog box functions launch processes. For example, clicking the OK button in the Array dialog box launches the process Execute Paste Array process.

Objects

Object (for object primitive, or primitive) is a generic term that refers to the individually editable items that are the building blocks in your design, such as a wire or a junction.

Connector components (Schematic 3)

Schematic 3 allows the user to define a special component type: Connector. Connector components (usually a single bitmap part) are a special catagory of multi-part component, where each placed "part" represents an individual pin with a common designator. This feature is not supported in Advanced Schematic.

To acheive the effect of "connectors" simply place a port, linked to the component and pin number (located elsewhere on the sheet). This allows the same flexibility with pin location across a single (or multiple) sheets.

Local hidden pins

One problem with hidden pins is that all hidden pins are deemed to be "global" when a netlist is generated. In other words, all hidden pins labeled VCC on all sheets are connected together in a common net. This can be a problem if the designer intends to split these supply nets. In simple cases, the designer can merely un-hide and manually wire the individual pins. However, this can quickly grow into a tedious task, in complex, hierarchical designs.

To overcome this limitation, the user can place an "unspecified" type port on the sheet, wired with a net label which matches the hidden pins net (e.g. VCC). The port should be given a unique name. This will make all pins with the same label "local" to the sheet. These pins will not be connected to other hidden nets with the same names on other sheets.

LIBEDIT.EXE (OrCAD)

Advanced Schematic Library Editor Application.

To create, edit and manage your component libraries.

Choose Edit Library from the Library menu. You can also launch the Library Editor as a separate application from the Windows Program Manager.

Copy (or Move) Component

This command allows you to copy (or move) a component from one library to another. When you choose this command, a dialog box opens, listing are current active libraries. Choose from one the the available library names and the component will be copied to, or moved to the destination library.

New Features for this Release

A number of new commands have been added to the Library Editor. Select a topic for information about these features:

Commands

Undo Limit option

This option, under Options Preferences, allows the user to set the number of Undo/Redo levels. The default is 50. This allows the user to backtrack through (and restore) 50 individual operations. The legal range is 0-16000. Because each operation must be stored in memory to enable Undo/Redo, setting a higher number may result in performance deterioration, while editing. If you notice Advanced Schematic slowing down after using the application for a while, try selecting a lower number. The more memory you have installed in your machine, to higher the limit you can set without hurting the performance of your system.

New Features list

Using Wildcard search fields

Wildcards can be used to define text strings when global editing all primitives that have text fields. This applies to parts,net labels, annotations (single line text), sheet symbols, sheet entries, ports and power ports.

Global text fields

When the Change dialog box for these primitives is opened, pressing the Global button opens the global editing options for that primitive. Three text entry boxes are displayed for each text field. The leftmost box shows the current text information for the item. The center field (under Attributes to Match By) is labeled Wildcard. The rightmost field is under Copy Attributes.

Syntax for Wildcard searches

The center field (Wildcard) defines which strings will be edited. If * is displayed (the defaualt), all strings for this field are available to be globally edited. This can be limited by defining specific cases, for example S* will limit the fields to strings beginning with S, etc. Wildcards are case in-sensitive.

The Copy Attributes (right) field defines the changes to be made to the string. Defining the change follows this syntax: {oldtext=newtext}. This means change portion of the string "oldtext" to "newtext". You can use multiple sets of brackets to define complex replacements. In this case the leftmost replacement is made, then the next on, etc. Although this is very powerful, you must take care, because the first change can effect subsequent replacements, possibly generating an unexpected result. Any mistakes can be corrected with the Undo command, however. You can further limit the replacement by typing {!Text=text} to make the changes case sensitive. In this case, "Text" becomes "text". Otherwise replacement is case in-sensitive by default.

New Features list

Run command options

The Tools menu allow the user to execute other applications directly. This allows the user to avoid using the Windows run command or searching through the Program Manager to launch other programs.

File Manager, Control Panel, Calculator, Clock and Notepad are default options.

Tools Setup is also used to assign Editors to three Advanced Schematic processes:

Text Editor: assigns the editor for editing and viewing report files;

Picture Editor: assigns the editor for placed graphic images;

CSV Editor: assigns the editor for CSV format Bill of Materials files.

Tools Setup is also used to assign links to other EDA applications and four additional User Commands.

Run command setup
New Features list

Protel (DOS) library conversion

Advanced Schematic uses vector, rather than bitmap graphics to display parts. A special library, PROTEL.LIB contains substitute vector parts for all bitmap components from the standard libraries delivered with Protel Schematic 3.3 (the DOS version).

When you load Schematic 3.3 sheet files, these components will be substituted automatically for the bitmap versions.

When a user-created component bitmap shape cannot be matched to PROTEL.LIB, Advanced Schematic will automatically "vectorise" these parts and report them as "unmatched." In many cases, the automatically vectorised components are perfectly acceptable, however, in some cases you may wish to improve their vector rendering.

The follow procedure can be used to edit these un-matched components and add these additional parts to PROTEL.LIB so that they can be automatically substituted the next time you load a Schematic 3 sheet file.

1. In the Schematic (sheet) Editor, use Library Add to open the special PROTEL.LIB, which should be in the same directory as the Advanced Schematic applications. PROTEL.LIB parts are named using a special Bitmap ID# which "describes" the orginal bitmap shape in the DOS version.
2. Open your Schematic DOS sheet file. Any components which cannot be matched (based upon shape) will be reported after the sheet is loaded. A text file named (filename).$$$ listing un-substituted parts will be generated and displayed using the default text editor (see under Run command). Leave this text file open -- you will need the information in this file later in this procedure.
3. Use the Library Make Project Library command to create a new library from the open sheet (or sheets). This library will include all project parts -- both substituted and non-substituted. Make sure that you save the library in Binary (.LIB) format.
4. Launch the Schematic Library Editor.
5. Open both your Project library AND the PROTEL.LIB library.
6. Using the text report, search the project library for the unmatched components. Click in the Components list box to display the first unmatched part. Advanced Schematic will have "vectorised" this part, substituting vectors, where possible for the bitmap patterns. Although some patterns may not be completely or optimally vectorised, it is a simple matter to manually clean-up the component, if desired, using the standard graphical editing tools in the Library Editor.
7. Using Rename Component, name the component using the Bitmap ID# in the report text, including the minus (-) sign, if any.
8. Using the Component Copy or Move command, add the renamed component into the PROTEL.LIB library.
9. Repeat this process for each unmatched component, being careful to rename the component with the precise Bitmap ID# in the report.
10. Save the PROTEL.LIB library, which now includes your newly vectorised parts. You can discard the project library.

When loading any subsequent Schematic 3 files, parts which match the (now cleaned-up) original bitmap version will be automatically substituted. So, it is only necessary to perform one "clean-up" per unique pattern.

New Features list

Edit File Types

File Types allows the user to assign a new file mask. For example, you may wish to change the default extension to search for other file types.

File masks are non-restrictive when opening files. Advanced Schematic identifies file types independently of the extension and converts other file types into Advanced Schematic formats automatically.