Commands

Run User Program 4
Run Windows Setup
Save Environment Configuration File
Screen Redraw
Search For Net
Select All
Select Connection
Select Inside Area
Select Net
Select Outside Area
Send To Back
Send To Back Of
Set Location Mark 10
Set Location Mark 2
Set Location Mark 3
Set Location Mark 4
Set Location Mark 5
Set Location Mark 6
Set Location Mark 7
Set Location Mark 8
Set Location Mark 9
Set Location Mark 1
Set Template File Name
Setup Autopan
Setup Printer
Setup Run Options
Sheet Options
Shift Cursor Down
Shift Cursor Left
Shift Cursor Right
Shift Cursor Up
Toggle Command Status Bar
Toggle Component Browser
Toggle Drawing Toolbar
Toggle Electrical Grid
Toggle Main Toolbar
Toggle Project Manager
Toggle Scroll Bars
Toggle Selection
Toggle Single Object
Toggle Snap Grid
Toggle Status Bar
Toggle Visible Grid
Toggle Wiring Toolbar
Undo
Up Hierarchy
Update Current Template
Update Parts
Using Help
View Read Only Part Fields
Window Arrange Icons
Window Cascade
Window Close All
Window Tile
Zoom 100%
Zoom 200%

File Open Sheet

Summary Open and load a schematic worksheet file.

To Launch F O - 

Purpose The file open command is used to load and display the contents of a schematic worksheet file into a new document window, ready to visually check or edit. You can load any number of schematic worksheet files into separate windows.

Outcome Once the schematic worksheet file has been selected it will load into a new document window and all the sheet preferences will be restored. This is because Advanced Schematic stores the sheet options in the worksheet whenever you save the file.

Comments Advanced Schematic can load any number of schematic worksheet files into separate windows using the Multiple Document Interface (MDI). Advanced Schematic will load old Protel Schematic 2.xx and 3.xx files as well as OrCAD STD III and STD IV.

When no document file is open, the Menu bar displays four options: File, Info, Library 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 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

File Open Project
Opening Schematic Files

File Open Project

Summary Open and load all schematic sheets in a project.

To Launch F J

Purpose The Open Project command is used to load and display the contents of all the hierarchical worksheet files contained in a schematic project.

Outcome Once the schematic project file has been selected, all the hierarchical schematic worksheet files will load into new document windows. The Project Manager will be displayed at the left side of the schematic application window, listing all the currently loaded schematic files.

Comments The Project Manager may be toggled off by clicking on the Project Manager button on the Main toolbar or by choosing Project Manager from the Options menu.

Procedure To open a schematic project:

1. Choose Open Project from the File menu.

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

3. Click OK to open the project file.

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

See also

File Open Sheet
Opening Schematic Files

File Save Project

Summary Save all the sheets in the current project.

Outcome When you select the File Save Project command, the contents of the current schematic project will be saved to the Advanced Schematic binary file format using the existing paths and filenames. A backup file is created for each of the sheets in the project, with ".bak" extension.

Comments If the schematic project 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.

Procedure To save the current schematic project:

1. Choose Save Project from the File menu.

See also

File Save All
File Save As
File Save
Save and Save As information

File New

Summary Create a new, empty schematic worksheet.

To Launch F N

Purpose The File New command is used to create a new blank schematic worksheet.

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

Comments You can change and define various sheet settings, such as sheet styles, size, orientation, color and other options that apply to the worksheet by choosing Sheet from the Options menu.

Procedure To create a new schematic worksheet:

1. Choose New from the File menu.

See also

Sheet Options

File Save All

Summary Save all loaded schematic project and worksheet files.

To Launch Alt F L

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

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

Procedure To save all schematic worksheets which are currently opened:

1. Choose Save All from the File menu.

See also

File Save
File Save As
File Save Project
Save and Save As information

File Exit

Summary Quit from Advanced Schematic.

To Launch F X

Purpose The File Exit command is used to close down all opened schematic worksheets, and end the current work session of Advanced Schematic.

Outcome When the File Exit command is selected, the current session of Advanced Schematic ends and the program prompts you to save modified files.

Procedure To quit the current session of Advanced Schematic:

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 file(s). Choose Yes to save the file(s), choose No to exit without saving the file(s) or 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 Advanced Schematics command processes, such as "Place Wire," "Select Net," "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 Advanced Schematic 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 button such as 'Right Click' from the Keys window. You can also use a combination of the CTRL and SHIFT keys.

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

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

See also

Hot Key Information

Preferences

Summary Setup System Preferences.

To Launch O P

Purpose The Options Preferences command is used to define various system settings, such as cursor shape, default template file, grid display type, selection color, auto-junction, dialog text font and other options that apply to all document windows.

Outcome The following are preferences 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. Advanced Schematic 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. Advanced Schematic will display all the available colors that your graphics card and monitor supports.

Cursor Type - Three options are available for the physical (or sheet) cursor, a large 90 degree cross, small 90 degree cross or a 45 degree cross. The sheet cursor is displayed when making selections, moving or placing objects. The sheet 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.

Project Manager Colors - The default colors of the Top, Real and Complex sheets that are displayed in the Project Manager panel may be changed at any time by clicking in the corresponding color box.

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 Schematic Editor must be closed and re-opended in order for this new font to change.

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

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.

Add Template to Clipboard - When this option is enabled, the current sheet template including border, title block, and any additional graphics will be copied to the Windows clipboard when the Copy or Cut command is used. This option only affects the Windows clipboard where objects are stored in WMF format. The sheet template is not added to the Advanced Schematic internal clipboard.

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.

Drag Orthoganal - When this option is enabled, dragging electrical objects will enforce wires to remain at 45/90 placement angle modes. Any angle or ruberbanding wire placement is used if this option is disabled. The spacebar can be used at anytime while moving objects to toggle through the 45/90/any angle placement modes.

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.

OrCAD Ports - When this option is enabled, the length of all the Ports in a schematic design/project are re-sized to the OrCAD equivalent and the length of the Port is restricted from being manually edited.

Copy OrCAD Footprint From/To - This option is used to map an OrCAD part field to Advanced Schematic Footprint field. This mapping is bi-directional, ie. used when loading and saving OrCAD files.

Object's Snap-to-Cursor Point - If the Center of Object option is enabled the cursor will jump to the center of an object when it is moved. If the Object's Electrical Hot Spot is enabled the cursor will jump to the nearest electrical hot spot when it is moved.

Default Template File - Specifies the default template (.DOT) file to be loaded when the File-New command is chosen.

Browse button - Allows user to browse available template (.DOT) files.

Procedure To setup the system preferences:

1. Choose Preferences from the Options menu.
2. Choose or type in new system settings.
3. Click OK to accept the new settings.

See also

Sheet Options
Preference Information

Toggle Component Browser

Summary Turn the Component Browser on or off.

To Launch O B - 

Purpose The Toggle Component Browser command is used to turn the Component Browser on or off at the left hand side of the Advanced Schematic application window.

Comments The Component Browser allows browsing of component names, placement of parts onto the current worksheet, renaming of part designators and jumping to a part that has been placed on a worksheet in a project. Refer to the User Guide for detailed descriptions about the Component Browser.

Procedure To turn the Component Browser on or off:

1. Choose Component Browser 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 Wiring Toolbar
Toggle Drawing Toolbar

Toggle Wiring Toolbar

Summary Turn the Wiring Toolbar on or off.

To Launch O W - 

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

Outcome Choosing Wiring Toolbar from the Options menu will turn the Wiring Toolbar on or off.

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

Procedure To turn the wiring toolbar on or off:

1. Choose Wiring Toolbar from the Options menu.

See also

Toggle Main Toolbar
Toggle Drawing Toolbar

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, towards 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 Wiring Toolbar

Toggle Status Bar

Summary Turn the Status Line on or off.

To Launch O T

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

Outcome Choosing Status line from the Options menu will turn the Status line on or off. This can also turn on/off the Command Status Bar (if it is toggled on in the Options menu).

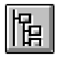
Comments The status line displays information about the current activity or mode. When a command is selected, the left side of the status line briefly describes the chosen command. The left side of the status line also indicates the current cursor position.

Procedure To turn the Status line on or off:

1. Choose Status line from the Options menu.

Toggle Project Manager

Summary Turn the Project Manager on or off.

To Launch O J - 

Purpose The Toggle Document Manager command is used to turn the Project Manager on or off at the left hand side of the Advanced Schematic application window.

Comments The Project Manager lists all the currently loaded schematic files and shows the hierarchical file structure of the project. The Project Manager is used to visually navigate through sheets in a flat-sheet or hierarchical project. You can view a schematic worksheet by clicking on a sheet within the Project Manager. Refer to the User Guide for detailed descriptions about Project Management and the Project Manager.

Procedure To turn the Project Manager on or off:

1. Choose Project Manager from the Options menu.

Popup Location Mark Menu

Summary Popup the Place Location Marks menu.

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

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

Procedure To pop up the Place Location Marks menu at the current cursor position.

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

See also

Hot Keys

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. Press the E keyboard hot key. (Default hot key assignment)

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. Press the P keyboard hot key. (Default hot key assignment.)

See also

Hot Keys

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

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.

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

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

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

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.

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

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

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

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.

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

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

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

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.

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

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

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

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.

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

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

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

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.

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

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

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

See also

Hot Keys

Popup Library Menu

Summary Popup the Library menu.

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

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

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.

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

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

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

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.

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

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

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

See also

Hot Keys

Information System Status

Summary Display system information.

To Launch I S

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

Comments System Status Information

Disk Usage

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

Free Disk Space - amount of disk storage space available on your hard disk in MegaBytes.

Virtual Memory Swap file - size of your virtual memory swap file. Use the Control Panel to change the virtual memory settings.

System Resources

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

Free Memory

Held by Windows - total free memory Windows is holding (in kilobytes), includes physical and virtual memory.

Held by this application - total free memory Advanced Schematic is holding (in kilobytes), includes physical and virtual memory. This memory is released back to Windows when Advanced Schematic application is closed.

Total memory - total free memory (in kilobytes), includes physical memory and virtual memory.

Physical memory - amount of physical memory (in kilobytes) your system currently has free, does not include virtual memory.

Date/Time

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.

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 Advanced Schematic.

To Launch H A

Purpose The Help About command is used to display the application name, version number and copyright information. This command can be used to find version number of Advanced Schematic before calling technical support.

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

Procedure To display the version number of Advanced Schematic:

1. Choose About from the Help menu.

See also

Help Index

Help Basic Concepts

Summary Basic information about Advanced Schematic.

To Launch H B

Purpose The Help Basic Concepts menu option displays an index of basic operational topics of Advanced Schematic, such as the design environment, system options etc.

Outcome Once the Help Basic Concepts command is selected the Advanced Schematic 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 Advanced Schematic index of menu commands, the topics offer access to detailed information about each command.

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

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 Advanced Schematic 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.

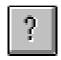
2. Then choose the specific topic from the help index list.

See also

Help Index

Help Index

Summary Help system topic index.

To Launch F1 - H C - 

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 Advanced Schematic 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 OrCAD

Summary Command Cross Reference for OrCAD SDT users.

To Launch H O

Purpose The Help OrCAD menu option displays the command cross-reference for OrCAD STD users.

Outcome Once the Help OrCAD command is selected the Advanced Schematic on-line help system is loaded and displays a cross-reference of command topics. Refer to the User Guide and reference supplement for information about OrCAD compatibility.

Procedure To display the OrCAD command cross-reference:

1. Choose OrCAD from the Help menu.

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

See also

Again
Block
Conditions
Delete
Edit
Find
Get
Hardcopy
Inquire
Jump
Library
Macro
Place
Quit
Repeat
Set
Tag
Zoom

Help Printing

Summary Information about generating hard copy output.

To Launch H P

Outcome Once the Help Printing command is selected the Advanced Schematic 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
Generating print or plot
Help Index

Help Reference

Summary Advanced Schematic Reference.

To Launch H R

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

Outcome Once the Help Reference command is selected the Advanced Schematic 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 schematic 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 schematic 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 schematic 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, non-minimized schematic 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 schematic 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 schematic document windows:

1. Choose Cascade from the Window menu.

See also

Window Tile

Window Arrange Icons

Summary Arrange all minimized open schematic windows.

To Launch Alt W I

Purpose The Window Arrange Icons command is used to arrange the icons of all minimized schematic documents across the bottom of the Advanced Schematic application window.

Procedure To Arrange all the minimized schematic document icons:

1. Choose Arrange Icons from the Window menu.

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 document windows and icons.

Outcome Once the Close All command has been chosen all the opened document windows and icons will be closed. If any document window has been modified since the last save a dialog box will be displayed and prompt you to save the file(s).

Procedure To close all the opened document windows and icons:

1. Choose Close All from the Window menu.

2. If any document window has been modified since last saving, a dialog box will be displayed that prompts to save the file(s). Choose Yes to save the file(s), No to close the file(s) without saving or choose Cancel to ignore the Close All command.

See also

File Close

Run Library Editor

Summary Switch to the Schematic Library Editor.

To Launch L E - 

Purpose The Run Library Editor command is used to switch to the Schematic Library Editor. The Schematic Library Editor allows creation, management, browsing and placement of components.

Outcome Launches the specified Library Editor.

Comments User can specify the Library Editor application under the Tools-Setup command. If the Library Editor has been previously loaded then the focus will switch to the Library Editor application window, otherwise the Library Editor will be launched, automatically loading previously opened Library files.

Procedure Choose the Library command from the Tools menu.

See also

Edit Component

Add/Remove Library

Summary Add and remove libraries from the library list.

To Launch L A - 

Purpose The Add/Remove Libraries command is used to add or remove libraries from the library list.

Outcome Once the library file has been selected it will appear in the Current File List window. All the added libraries will appear in the library list box in the Component Browser Panel.

Comments The library components are not loaded into memory, only their names. Therefore, an unlimited number of libraries may be added to this list.

Procedure To add or remove libraries from the library list:

1. Choose Add/Remove from the Library menu, or click the Add/Remove button in the Component Browser panel.

2. Type the library name or wild card mask (e.g. *.LIB) in the file name box.

3. Click OK to list the file(s) in the Files window of this dialog box.

4. Select a file and then click the Add button to add the library to the list.

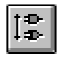
You can also double-click on the desired filename in the Files window. To change directories, click under any of the options listed in the Directories window.

See also

Adding libraries to the current list

Increment Part Number

Summary Toggle part number on multi-part component.

To Launch Alt E I - 

Purpose The Increment Part Number command is used to visually toggle through the available part numbers for a specific multi-part component.

Outcome As you click on the part its next part number is displayed.

Procedure To toggle through the part numbers:

1. Choose Increment Part Number from the Edit menu.
2. Position the cursor over a part on the worksheet and click ENTER or LEFT MOUSE.

Place Part from Browser

Summary Place a part from the Component Browser.

Purpose The Place Part from Browser command is used to place the currently highlighted part in Component Browser. The command is represented by the Place button on the Browser.

Comments You can also place parts from the Library Editor or by using the Place Part command.

Procedure To Place a part from the Component Browser:

Make sure that the part you wish to place is highlighted in the Components In Library window on the Component Browser.

1. Press the Place button on the Component Browser. The part will then be displayed in the workspace.

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.

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

See also

Place Parts

Edit Component

Summary Switch to the Schematic Library Editor and edit component.

Purpose The Edit Component command is used to switch to the Library Editor and edit the component currently displayed in the schematic editor Component Browser. The command is represented by the EDIT button on the Browser.

Outcome The focus will switch to the Library Editor application and the component will then be displayed in the workspace.

Comments To reflect changes made to the component, save the library and then select Update Schematic from the library editor File menu.

Note: All the sheets currently opened will be affected by this.

Procedure To edit a component from the Component Browser:

Make sure that the Advanced Schematic Library Editor is already running and the part you wish to edit is highlighted in the Components In Library window on the Component Browser.

1. Press the Edit button on the Component Browser.

See also

Run Library Editor

Rename Object Text

Summary Changes the text field associated with an object from the Browser.

Purpose Changes the designator of the current part (or text associated with other objects) in the Browser. The command is represented by the TEXT button on the Browser.

Outcome The new object text field will be listed within the Browser.

Procedure To change an object's text, first make sure that the object you wish to rename is highlighted in the Browser.

1. Press the Text button on the Browser.

2. Type the new text into the dialog box and press ENTER or click OK.

Jump to Object

Summary Jump to an object within a single sheet or project.

Purpose The Jump button in the Browser allows you to conveniently locate a specific object that has been placed within a schematic project, without having to zoom, pan or scroll through multiple screens or windows.

Outcome The object will be positioned in the center of the schematic document window and be placed in focus. The current window focus will change if the part is not on the current active document window.

Procedure To Jump to an object:

Make sure that the object type you wish to jump to is selected in the Browser and that the object name is highlighted in the Browser list box.

1. Press the Jump button on the Browser.

See also

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 schematic drawing 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 schematic sheet.

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 Document or All Documents. Single Document, prints the current document window, All Documents, prints all the schematic sheets in the current project.

Color Mode - Allows you to choose between Color or Monochrome modes. Color mode allows you to print the schematic sheet(s) in color, if you are using a color-capable output device supported by a Windows 3 driver. Monochrome will print the schematic 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 schematic sheet by specifying a scale factor. If you wish to enlarge the sheet by 200% set the Scale to 2.000. The default value is 1.000.

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

Comments Advanced Schematic 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 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 update the right portion of the dialog box, showing the orientation, positioning, margins and number of pages, before sending the file to the printer.

See also

__File Print__

Generating a print or plot

File Close

Summary Close the current window.

To Launch Alt F C

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

Procedure To close the current document window:

1. Choose Close from the File menu.

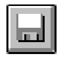
2. If the schematic document window has been modified since last saving, a dialog box will be displayed that prompts to save the file. Choose Yes to save the file, No to close without saving or choose Cancel to ignore the Close command.

See also

Window Close All

File Save

Summary Save current schematic worksheet with same file name.

To Launch Alt F S - 

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

Outcome When you select the File Save command, the contents of the current schematic document window will be saved to the Advanced Schematic binary file format using the existing paths and filenames.

Comments If the schematic worksheet 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 schematic worksheet:

1. Choose Save from the File menu.

See also

File Save All
File Save As
File Save Project

File Save As

Summary Save current schematic worksheet with new file name.

To Launch Alt F A

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

Outcome When you select the File Save As command, the Save Schematic 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.

Comments There are three file format options:

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

Protel Text Protel's published (ASCII) text format for schematic 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 Advanced Schematic, it will be automatically saved in the default binary format when you choose File Save.

OrCAD This format allows you to export an Advanced Schematic file into OrCAD STD IV format. When you load an OrCAD file into Advanced Schematic, it will be automatically saved in the default binary file format when you choose File Save. If you save your files in OrCAD format you should be aware of the limitations that OrCAD database imposes on Advanced Schematic data objects, in particular, most of the drawing tools including, elliptical arcs, ellipses, pie charts, rounded rectangles, beziers and graphic images will not be included in the OrCAD format files. Advanced Schematic also includes special attributes not supported by OrCAD, such as color for individual objects, True Type fonts, Port length etc.

Refer to the User Guide and on-line help for complete descriptions on Orcad compatability.

Procedure To save the current schematic 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 schematic worksheet. To change the output format of the file, choose one of the options listed in the File Format window.

3. Click OK to save the file.

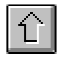
To change directories, click under any of the options listed in the Directories window.

See also

File Save All
File Save
File Save Project

Up Hierarchy

Summary Switch to the parent sheet of the current sheet.

To Launch Alt F H U - 

Purpose The Up Hierarchy command is used to move the focus to the next level up the design hierarchy from the current, active window.

Outcome Once the Up Hierarchy command is selected you are prompted to choose one of the sheet symbols, once a sheet symbol has been selected the focus moves to the next level up the design hierarchy from the current, active window.

Comments To move up hierarchy the top level sheet needs to be already open.

Procedure To move up the hierarchy structure:

1. Choose Up Hierarchy from the File-Hierarchy menu.

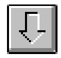
2. Position the cursor over a Sheet Symbol and click LEFT MOUSE or ENTER.

See also

Down Hierarchy
Sheet Symbol

Down Hierarchy

Summary Switch to a child sheet of the current sheet.

To Launch Alt F H W - 

Purpose The Down Hierarchy command is used to move the focus to the next level down the design hierarchy from the current, active window.

Outcome Once the Down Hierarchy command is selected you are prompted to choose one of the sheet symbols, once a sheet symbol has been selected the focus moves to the next level down the design hierarchy from the current, active window. If the child sheet is not already opened Advanced Schematic will automatically open the sheet.

Procedure To move down the hierarchy structure:

1. Choose Down Hierarchy from the File-Hierarchy menu.

2. Position the cursor over a Sheet Symbol and click LEFT MOUSE or ENTER.

See also

Up Hierarchy
Sheet Symbol

Annotate

Summary Re-Designate all Parts in schematic project.

To Launch Alt F T

Purpose The Annotate command is used to rename all the Parts on a schematic worksheet or Project so they are unique and in consecutive order.

Outcome Once the Annotate command is selected all Parts in your schematic design will be re-designated.

Comments Refer to the on-line help system and the reference supplement for further information about the Annotate command. Annotate will re-designate all the parts in the opened sheets in the current project.

Procedure To Annotate all Parts in a schematic design:

1. Choose Annotate from the File menu.

See also

Back Annotate

Back Annotate

Summary Back Annotate Part and Pin information from a PCB Was/Is File.

Purpose The Back Annotate command is used to update the schematic component designators and pin information from a PCB Was/Is file.

Comments Refer to the on-line help system and the reference supplement for further information about the Back Annotation command. Applies to the entire project, all the sheets used in the project need to be opened, at the time of back annotation.

Procedure To Back Annotate from a PCB Was/Is file:

1. Choose Back Annotate from the File menu.

2. Type the name of the PCB Was/Is file.

3. Click OK to change the schematic design.

See also

Annotate

Bill of Materials

Summary Generate a Bill of Materials.

To Launch Alt F R B

Purpose The Report Bill of Materials command is used to generate an ASCII parts list file of the Parts (including quantities) used in a schematic design.

Outcome Two BOM formats are generated: The CSV (comma separated value) format includes all sheet-level and library part text fields for each instance of a part. The condensed Protel (ASCII) format that displays the designator, type and count for each part.

Comments The CSV file format can be loaded directly into a spreadsheet or database application.

Procedure To generate a Bill of Materials file:

1. Choose Reports Bill of Material from the File menu.

See also

Report Project Hierarchy
Report Cross Reference

Report Project Hierarchy

Summary Generate a text listing to represent the design hierarchy.

To Launch Alt F R P

Purpose The Report Hierarchy Tree command is used to generate an ASCII file of the hierarchy tree structure used in a schematic design.

Outcome Once the Hierarchy tree command is selected a dialog box will open prompting to select the type of format for the output file, once the format has been chosen, a second dialog box will open asking for an output file name. Type a file name and click OK to save the hierarchy tree file.

Comments The CSV file format should be used when exporting the hierarchy tree file into a spreadsheet or database application.

Refer to the on-line help system and the reference supplement for further information about the Report Project Hierarchy command.

Procedure To generate a hierarchy tree file:

1. Choose Project Hierarchy from the File menu.

2. Select the type of format for the hierarchy tree file (CSV or Protel) then click OK.

3. Type a file name for the Hierarchy Tree file.

4. Click OK to generate the file.

See also

Bill of Materials
Report Cross Reference

Report Cross Reference

Summary Generate a part/sheet cross reference for the current project or sheet.

To Launch Alt F R C

Purpose The Report Cross Reference command is used to generate an ASCII file of the part/sheet cross reference used in a schematic design.

Outcome Once the Cross Reference command is selected a dialog box will open prompting to select the type of format for the output file, once the format has been chosen a second dialog box will open asking for an output file name. Type a file name and click OK to save the Cross Reference file.

Comments The CSV file format should be used when exporting the Cross Reference file into a spreadsheet or database application.

Refer to the on-line help system and the reference supplement for further information about the Report Cross Reference command.

Procedure To generate a Cross reference file:

1. Choose Cross reference from the File menu.

2. Select the type of format for the Cross Reference file (CSV or Protel) then click OK.

3. Type a file name for the Cross Reference file.

4. Click OK to generate the file.

See also

Bill of Materials
Report Project Hierarchy

Undo

Summary Undo previous command.

To Launch Alt BackSpace - Alt E U - 

Purpose The Undo command is used to restore a document 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 Advanced Schematic 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. User sets the limit of Undo/Redo steps under Options-Preferences. If set to 0, then the stack is cleared, releasing additional memory. Does not support Annotate or BackAnnotate commands.

Note: When using Undo/Redo after complex cross-project global edits, only the current sheet (in focus) will be affected.

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.

Comments Advanced Schematic 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. User sets the limit of Undo/Redo steps under Options-Preferences. If set to 0, then the stack is cleared, releasing additional memory. Does not support Annotate or BackAnnotate commands.

Note: When using Undo/Redo after complex cross-project global edits, only the current sheet (in focus) will be affected.

Procedure To redo the last operation:

1. Choose Redo from the Edit menu.

See also

Undo

Electrical Rules Check

Summary Perform an Electrical Rules Check on the current project or sheet.

To Launch Alt F R E

Purpose The Electrical Rules Check command is used to automatically check the logical integrity of your schematic design. This feature should be used prior to generating a netlist.

Outcome Depending upon the options specified, running ERC can produce two results. First, a text report listing of electrical and logical violations and warnings for the current active sheet, hierarchical or multi-sheet flat project. Secondly, error markers are placed in sheets at the site of specific ERC violations as an aid in tracking and correcting reported problems.

The following are options that can be set within the Electrical Rules Check dialog box.

Multiple Net Names On Net

Reports physical nets with net identifiers with different names.

Unconnected Net Labels

Reports net labels that are not physically connected to at least one other electrical item in the sheet.

Unconnected Power Objects

Reports power objects that are not physically connected to at least one other electrical item.

Duplicate Sheet Numbers

Reports sheets that have been assigned the same sheet number(Options-Sheet dialog, Document Info option).

Duplicate Component Designators

Reports parts that have identical designator labels. This condition can occur when the Annotate command has not been used or when the File-Hierarchy-Complex to Simple command has not been used to "flatten" a complex hierarchical project (project with duplicate sheets).

Bus Label Format Errors

Reports net labels attached to buses which are not legally formatted to reflect a range of signals. Logical connectivity for buses can be assigned by placing a net label on the bus. Generally, this net label will include all bus signals, e.g. HA[0..19] represents nets named HA0, HA1, HA2, etc. to HA19.

Floating Input Pins

Reports unconnected part pins whose function is identified as "Input."

Suppress Warnings

This option generates a report and error markers for error conditions only. Warning conditions (see Pin / Sheet Entry / Port Rule Matrix, below) are ignored. This allows the designer for perform quick ERCs for all error-level problems.

Create Report File

This option generates a text report listing all ERC report information.

Add Error Markers

This option generates special error markers in sheets, at the site of each reported warning or error. Special facilities in the sheet editor allow the user to jump from error marker to error marker.

Descend Into Sheet Parts

This options treats sheet parts as hierarchical sheet symbols. A sheet part is a part whose unconnected pins are associated with ports on sheet which descends hierarchically from the sheet part. The "descending" sheet is defined in the part's Sheet Part path field.

Current Sheet Only

This option is used to restrict the check to be done on the current sheet or the entire project.

Net Identifier Scope

This option is used to define the use of net identifiers (net labels, ports and sheet entries) in the project.

Net Labels and Ports Global

With this option, net labels are assumed to apply to all sheets in a project. In other words, nets are global and each instance of a net name or port is deemed to be connected to all other identically named identifiers.

Only Ports Global

This option treats net labels as local only. All identically named nets on a sheet are deemed to be connected. Intersheet connections occur only through identically labeled ports. This model works like the OrCAD SDT "flat" project model.

Sheet Symbol / Port Connections

This option makes intersheet net connections only through sheet symbol entries and subsheet ports. Ports are deemed to be connected only to identically named nets entering through their sheet symbols on parent sheets. This model works like the OrCAD SDT "hierarchical" project model.

Pin / Sheet Entry / Port Rule Matrix

An error or warning can be specified using a matrix of pin, port or sheet entry connection or non-connection conditions. For example, connected input pins would not normally be regarded as an error condition but connected output pins would be regarded as an error. The default values for each of these conditions reflects this. The user can specify either errors or warnings for connections of pin types, ports or sheet entries. Items which are set as warnings give the user the option to supress warnings when performing preliminary checks, which will keep these initial ERC reports short and manageable. Warnings can then be comprehensively checked by the designer, later in the design validation process.

To change the default settings in the matrix, the user need only click LEFT MOUSE in any matrix square. With each click the square will toggle from No Report (green), Warning (yellow) and Error (red), then back to No Report, etc.

Comments No ERC symbols can be placed on pins and other electrical objects, such as un-connected net labels to suppress the ERC checking.

Procedure To perform an Electrical Rules Check on the current design or sheet:

1. Choose Reports and then Electrical Rules Check from the File menu.

2. Specify the ERC options for the project:

3. Click OK to generate the report file anderror markers.

See also

Error Marker
ERC Information

Select Connection

Summary Find physical connection between pins.

To Launch Alt E S C

Purpose The Select Connection command is used to select the free primitives (wires, junctions, net labels) that connect any two Part pins. This command is useful for identifying two Part pins that are physically connected and, therefore, will be processed when generating a netlist.

Outcome Once a wire, junction or some other electrical object has been chosen, the connection will be selected in the color that is defined for selections in the Options Preferences dialog box.

Comments The selected primitives that make up the connection can be cleared, copied or cut to the clipboard and then pasted into the current document window or to another schematic worksheet.

Procedure To physically select a connection:

1. Choose Select connection from the Edit menu.

2. Then click on a wire, junction or Part pin to select the connection.

See also

Select Net
Information Selected Pins

Select Net

Summary Find physical connection from a point.

To Launch Alt E S N

Purpose The Select Net command is used to select the free primitives (wires, junctions, net labels etc.) that connect a physical net. This command is useful to identify all the pins physically connected by wires or by net labels. To see a listing of all the selected pins use the Info Selected Pins command.

Outcome Once a wire, junction or some other electrical object has been chosen, the net will be selected in the color that is defined for selections in the Options Preferences dialog box.

Comments The selected primitives that make up the net can be cleared, copied or cut to the clipboard and then pasted into the current document window or to another schematic worksheet.

Procedure To physically select a Net:

1. Choose Select Net from the Edit menu.

2. Then click on a wire, junction or Part pin to select the Net.

See also

Select Connection
Information Selected Pins

Create Netlist

Summary Generate a netlist from the current project or sheet.

To Launch Alt F E

Purpose The Create Netlist command is used to generate an ASCII text netlist file in the Protel format, a large number of other netlist output formats are supported. The typical netlist format includes descriptions of Parts, such as the designator and package type combined with the pin-to-pin connections that define each net. Loading a netlist into a printed circuit board layout package automates many of the tedious and error prone operations inherent in the design process.

Outcome All netlist formats supported by Advanced Schematic are generated as ASCII text files that carry two types of information.

1. Descriptions of the Parts in the circuit.

2. A list of all pin-to-pin connections in the circuit.

Some netlist formats combine both sets of data in a single description, others, including the Protel format, separate the data into two sections.

Netlist formats

Algorex, AppliconBRAVO, AppliconLEAP, Cadnetix, Calay, Calay90, Case, CBDS, ComputerVision, EDIF 2.0, EDIF 2.0 Hierarchical, EEDesigner, EEsof Libra, EEsof Touchstone, FutureNet, Hilo, Integraph, Mentor BoardStation 6, Multiwire, Orcad - PLDnet, Orcad - PCB II, PADS Ascii, PCAD, PCAD NLT, Protel - Advanced PLD, Protel, Protel 2, Protel - Hierarchical, Protel Wirelist, Racal Redac, Scicards, Spice, Spice Hierarchical, Star Semiconductor, Tango, Telesis and Vectron.

Comments If you intend to manually edit a netlist, make sure that you save the results in an "unformatted" or "text only" form, as hidden formatting or control characters can render a netlist unreadable by Protel for Windows.

The "Action after netlist generation" option is used to launch a specified application (eg. a Digital Simulation package) after generating a netlist.

Procedure To generate a netlist file:

1. Choose Create Netlist from the File menu.

2. Select the netlist output format, the net identifier scope.

3. Click OK to generate and save the netlist file.

See also

Generating Netlists
Electrical Rules Check

Cut

Summary Copy selected objects to clipboard and remove from sheet.

To Launch Shift Delete - Alt E T - 

Purpose The Edit Cut command is used to clear the current selected objects from the worksheet and copies it to the clipboard. The Edit Paste command can be used to place the selection back into any open Advanced Schematic document window or to another application that supports the Windows .WMF clipboard format.

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. The Add Template to Clipboard option (Options-Preferences command) includes the sheet template (border and title block, etc.) along with the selection copied to the clipboard.

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. 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
Copy

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 worksheet to the clipboard. The Edit Paste command can be used to place a copy of the selection back into any open Advanced Schematic document window or into any application which supports the .WMF (Windows MetaFile) clipboard format.

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. The Add Template to Clipboard option (Options-Preferences command) includes the sheet template (border and title block, etc.) along with the selection copied to the clipboard.

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 th 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 (Options-Preferences) 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 worksheet.

To Launch Shift Insert - Alt E P - 

Purpose The Paste command is used to place the current clipboard contents into any open Advanced Schematic document window. Advanced Schematic has its own clipboard format. The standard Windows clipboard is not used when Pasting.

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 select a location 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 worksheet and click LEFT MOUSE or press ENTER.

See also

Cut
Copy

Clear

Summary Delete all selected objects from the current worksheet.

To Launch Ctrl Delete - Alt E L

Purpose The Clear command is used to remove selected objects from the worksheet without copying the selection to the clipboard.

Outcome The selection will be cleared from the display.

Comments You can use the Edit Undo command (shortcut: S8 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.

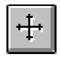
2. Choose Clear from the Edit menu. (shortcut: CTRL+DELETE).

See also

Undo
Cut
Delete
Delete Single Object

Move Selection

Summary Move selected objects to another area of the worksheet.

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 key press rotates the selection 90 degrees counter-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 worksheet:

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.

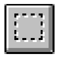
3. Position the selection in the worksheet and click ENTER or LEFT MOUSE.

See also

Move
Drag
Drag Selection

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 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 to "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 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 to "Select Second Corner."

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

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

See also

Select Inside Area
Select All

Select All

Summary Select everything on the current worksheet.

To Launch Alt E S A

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

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

Comments Select all will not select borders or title blocks but will include text frames or annotations within title blocks.

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 worksheet.

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 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.

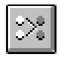
4. Press ENTER or LEFT MOUSE to complete the selection.

See also

De-Select Inside Area
De-Select All

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 schematic document window.

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 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 schematic worksheet 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 worksheet or when you wish to quickly add and remove a number of objects.

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

1. Choose Toggle Selection from the Edit menu.

2. Select or de-select the objects in the worksheet 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 on a schematic worksheet by moving the cursor over the object and clicking SHIFT + LEFT MOUSE.

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

Comments The Toggle 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 toggle the selection of a single object in an open document:

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
Hot Keys

Delete

Summary Select and delete objects on schematic worksheet.

To Launch Alt E D

Purpose The Delete command is used to delete objects on a schematic worksheet.

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.

Procedure To Delete objects in an open document:

1. Choose Delete from the Edit menu.

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

As you click on an object, it will be cleared from the worksheet and the status line will continue to prompt you to select 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 S8 LEFT MOUSE. The object will then be displayed with graphical editing handles.

3. Press the DELETE key.

When you press the DELETE key, the object will be cleared from the worksheet.

See also

Undo
Delete
Clear

Change Object Graphically or Set Focus

Summary Select and change objects on schematic sheet.

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 normally only be accessed using the mouse. This can be changed using Option-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 by clicking left mouse on a graphical handle and dragging that handle to a new location.

See also

Change

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 schematic worksheet. Each object or primitive has its own range of editable attributes. You can change one object or extend changes across your entire design using powerful global editing options.

Outcome Opens the object's dialog box, displaying all editable attributes.

Comments The system will not prevent changes that violate specific design rules. An Electrical Rule Check (ERC) should always be performed before generating a netlist.

Global ChangesChanges can be made to a single selected object or they can be applied globally across the entire worksheet using flexible, powerful global editing options. Virtually every editable attribute can be globally applied. A simple example would be changing the font style assigned to a single text string. Typically, the engineer would want this new font style applied to other strings as well. However, it might be important that the new font style be applied only to strings with the same size as the selected string, or perhaps strings of a specific style or size only, etc.

All of these options (and more) are supported by the Change command in Advanced Schematic. The possible applications for global changes are limited only by the imagination of the 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. Matching attributes for global changes can be assigned by clicking the Options button in any of the Change object dialog boxes. When you click Options, the dialog box expands to display the parameters for global matches.

Each Change object dialog box may contain different options since every type of object has a unique set of attributes. For example, junctions have five changeable attributes X-location, Y-location, size, color and selection. While Ports have eleven. For detailed descriptions of all objects and assigned attributes, see the Data Primitives section.

Global change options

Any attribute can be globally changed if the object's dialog box

includes a Match By and Copy field with the Attribute field.

Match By

There are three options for this field: Same (apply global

changes if this object attribute is matched in the target object);

Different (apply global changes if this attribute is not a match

in the target object) and Any (the default) which applies the change irrespective of whether the attribute has the same value in both objects.

Copy

If the Copy box is checked, then this attribute will be copied to the target object. These fields allow the designer to select a range of attributes that will be copied to other objects.

Change Scope

These three options define the extent of changes: Change This Item Only (no global changes); Change Matching Items In Current Document; Change Matching Items In All (currently opened) Documents. The latter option extends your changes across all open sheets, allowing you to globally edit an entire project.

Using wildcards when globally editing text

Many objects include text fields. These text fields can also be globally edited with the additional capability of using wildcards to define changes. This applies to parts, net labels, annotations (single line text), sheet symbols, sheet entries, ports and power ports. Separate Edit-Find Text and Edit-Replace Text commands allow text replacement across different object types. These commands also support the wildcard search and replace sytax, described below.

Syntax for wildcard search and replace edits

The Match By field 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 field for text strings is used to define the changes to be made to the string. Braces "" and "" are used to define the rules for text replacement. If this field is empty then the entire string will be replaced.

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.

Procedure 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:

1. Choose Edit-Change (shortcut: E, C);

The prompt "Select (item)" will appear on the status line.

2. 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.

See also

Change Object Graphically or Move
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 schematic worksheet. 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.

Comments The system will not prevent changes that violate specific design rules. An Electrical Rule Check (ERC) should always be performed before generating a netlist.

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

Procedure To change any placed object:

1. Position the cursor over the target object and double click LEFT MOUSE.

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

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

See also

Change

Move

Summary Select and move objects on schematic sheet.

To Launch Alt E M M

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

Comments Advanced Schematic automatically stacks objects, text and graphics in layers on a schematic sheet. 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.

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 pressing 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:

1. Choose Move, 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 select it. Use the Zoom options (or press PGUP) to enlarge your view of the layout 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. Select another object to be moved or press ESC or RIGHT MOUSE to leave the move command.

See also

Move Single Object

Move Single Object

Summary Select and move objects on the schematic worksheet.

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

Comments Advanced Schematic automatically stacks objects, text and graphics in layers. 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 select it. Use the Zoom options (or press PGUP) to enlarge your view of the layout 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 schematic worksheet. This command is similar to the Move command, except that when an object has been moved and placed on the worksheet it will be positioned in front of all the other stacked objects on the sheet.

Comments Advanced Schematic automatically stacks objects, text and graphics in layers. 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 sheet. Note: This feature has no connectivity significance.

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 worksheet:

1. Choose Move, Move To Front from the Edit menu. The prompt "Select object" will appear on the status line.

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 select it. Use the Zoom options (or press PGUP) to enlarge your view of the layout 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.

The status line will prompt "Select object". Select another object to be moved or

4. 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 schematic worksheet to the front of all other objects.

Comments Advanced Schematic automatically stacks objects, text and graphics in layers . 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 sheet. Note: This feature has no connectivity significance.

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

1. Choose Move, Bring To Front from the Edit menu. The prompt "Select object" will appear on the status line.

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

When the object has been selected it will be positioned in front of all the other objects on the sheet. Click on another object to bring it the front or;

3. Press ESC or LEFT 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 schematic worksheet to the back of all other objects.

Comments Advanced Schematic automatically stacks objects, text and graphics in layers. 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 sheet. Note: This feature has no connectivity significance.

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

1. Choose Move, Send To Back from the Edit menu. The prompt "Select object" will appear on the status line.

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

When the object has been selected it will be positioned at the back of all the other objects on the sheet. Click on another object to send it to the back or;

3. 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 schematic worksheet to the front of another object. This command is similar to the Bring To Front command, except that you will be prompted to indicate the object that the selected object will be placed in front of.

Comments Advanced Schematic automatically stacks objects, text and graphics in layers. 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 and click on an object it will be positioned at the bottom of the stack, behind all the other objects on the sheet. Note: This feature has no connectivity significance.

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

1. Choose Move, Bring To Front Of from the Edit menu. The prompt "Select object" will appear on the status line.

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

The object will temporarily disappear from the worksheet exposing any objects behind it. The prompT "Select object" will appear on the status line again.

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 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 schematic worksheet 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 Advanced Schematic automatically stacks objects, text and graphics in layers. 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 sheet. Note: This feature has no connectivity significance.

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

1. Choose Move, Send To Back Of from the Edit menu. The prompt "Select object" will appear on the status line.

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

The object will temporarily disappear from the worksheet exposing any objects behind it. The prompt "Select object" will appear on the status line again.

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

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

See also

Send To Back
Bring To Front

Place Arcs

Summary Place graphical arcs on the worksheet.

To Launch Alt P D A

Purpose The Place Arc command is used to place an arc on the worksheet using the center of the arc as the starting point. Arcs are used for adding reference information to a schematic design, such as building graphical or mechanical symbols.

Comments When using the Place Arcs 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 current worksheet:

1. Choose Drawing Tools and then Arcs from the Place menu.

While moving the arc, 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 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 Arcs

Place Polygons

Summary Place graphical polygon shapes on the worksheet.

To Launch Alt P D P - 

Purpose The Place Polygons command is used to place graphical polygon objects on the current schematic worksheet. Polygons can be used for adding reference information to a worksheet, such as building graphical symbols, custom sheet borders, and title blocks.

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

Procedure To place a polygon on the current worksheet:

1. Choose Drawing Tools and then 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.

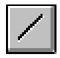
5. To exit this command press ESC or RIGHT MOUSE a second time.

See also

Polygon

Place Lines

Summary Place graphical lines on the sheet.

To Launch Alt P D L - 

Purpose The Place Lines command is used to place graphical lines on the current schematic worksheet. Lines are used for adding reference information to a worksheet, such as building graphical symbols, custom sheet borders, and title blocks.

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

Procedure To place a line on the current worksheet:

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 polyline.

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.

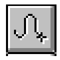
5. To exit this command press ESC or RIGHT MOUSE a second time.

See also

Line

Place Beziers

Summary Place bezier curves on the sheet.

To Launch Alt P D B - 

Purpose The Place Bezier command is used to place curved line segments on the current worksheet. Beziers are used for adding reference information to a worksheet, such as building graphical, mechanical or electrical symbols.

Comments When using the Place Beziers 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.

Remember, it takes four points to make a curve segment, (two control points and two end points). Therefore, if you add two points to an existing curve, you will not get a full segment.

Procedure To place a Bezier on the current worksheet:

1. Choose Drawing Tools and then 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 S8 LEFT MOUSE or pressing the INSERT key. If you make a mistake, you can press DELTE 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 Ellipses

Summary Place elliptical shapes on the sheet.

To Launch Alt P D E - 

Purpose The Place Ellipses command is used to place elliptical shapes on the worksheet using the center of the ellipse as the starting point. Ellipses are used for adding reference information to a schematic design, such as building graphical or mechanical symbols.

Comments When using the Place Ellipses 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 current worksheet:

1. Choose Drawing Tools and then Ellipses from the Place menu.

While moving the ellipse, 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 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.

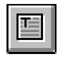
Start a new ellipse by selecting a center point or press ESC or RIGHT MOUSE to exit this command.

See also

Ellipse
Place Elliptical Arcs

Place Text Frame

Summary Place multi-line text frames on the sheet.

To Launch Alt P T F - 

Purpose The Place Text Frame command is used to place detailed notes or descriptive text on the worksheet. Uses might include placing revision history, circuit and logic documentation or some other description on the sheet.

Comments When using the Place Text Frame command the default placement field values are determined by the last placed text frame. Press TAB to change the default text, font style, text color, border width etc. during placement.

Procedure To place text on the current worksheet:

1. Choose Text and then Text Frame from the Place menu.

While Text Frames can be rotated or flipped along the x or y axis, this has no effect on the orientation of the text. Press TAB to manually change the default values of the text frame by typing new values directly into a dialog box.

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

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

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

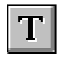
4. Place another text frame or press ESC or RIGHT MOUSE to exit this command.

See also

Text Frame
Place Annotation

Place Annotation

Summary Place single line text on the sheet.

To Launch Alt P T A - 

Purpose The Place Annotation command is used to place single line notes or descriptive text on the worksheet. Uses might include placing section headings, revision history, timing information or some other description on the sheet.

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

Procedure To place single line text on the current worksheet.

1. Choose Text and then Annotation from the Place menu.

While moving the annotation, 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 annotation by typing new values directly into a dialog box.

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

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

See also

Annotation
Place Text Frame

Place Pie Charts

Summary Place pie shapes on the sheet.

To Launch Alt P D C - 

Purpose The Place Pie Charts command is used to place pie shaped graphical objects (closed ellipses with angular cutouts) on the worksheet. Pies can be placed as reference information, used in building graphical symbols, custom sheet borders, or title blocks.

Comments When using the Place Pie Charts 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 current worksheet:

1. Choose Drawing Tools and then Pie Charts from the Place menu.

While moving the pie chart, 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 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.

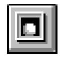
Place another pie chart or press ESC or SRIGHT MOUSE to exit this command.

See also

Pie Chart

Place Graphic

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

To Launch Alt P D G - 

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

Comments 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.

Procedure To place graphical objects onto the worksheet;

1. Choose Drawing Tools and then Graphic from the Place menu.

2. Type the image filename (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 filename in the Files window, if any. To change directories, click under any of the options listed in the Directories window.

While moving the graphic, press TAB to manually change the default values of the graphic by typing new values directly into a dialog box.

3. Position the cursor where the center of the text frame 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.

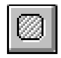
5. The Image file dialog box will be displayed again, choose another graphic image or press ESC or cancel to exit this command.

See also

Graphic

Place Round Rectangle

Summary Place round rectangles on the worksheet.

To Launch Alt P D O - 

Purpose The Place Round Rectangle command is used to place a rounded rectangle on the worksheet using the center of the rounded rectangle as the starting point. Rounded rectangles are used for adding reference information to a schematic design, such as building graphical or mechanical 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 current worksheet:

1. Choose Drawing Tools and then 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 pressing the 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 SESC or RIGHT MOUSE to exit this command.

See also

Round Rectangle
Place Rectangles

Place Elliptical Arcs

Summary Place elliptical arcs on the worksheet.

To Launch Alt P D I - 

Purpose The Place Elliptical Arcs command is used to place an elliptical arc on the worksheet using the center of the elliptical arc as the starting point. Elliptical arcs are used for adding reference information to a schematic design, such as building graphical or mechanical symbols.

Comments When using the Place Elliptical Arcs 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 current worksheet:

1. Choose Drawing Tools and then Elliptical Arcs from the Place menu.

While moving the elliptical arc, 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 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 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 Arcs](#)

Place Bus

Summary Place bus lines on the worksheet.

To Launch Alt P B - 

Purpose The Place Bus command is used to place a graphical Bus line on the current worksheet. A Bus line is used to represent a common pathway for multiple signals on a worksheet.

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

Buses can be attached to Ports or sheet symbols, for connection to other schematic sheets.

Procedure To place a Bus connection:

1. Choose Bus from the Place menu.

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

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

As you move the mouse or press the arrow keys, a highlighted Bus line segment will be displayed. You can change the Bus placement mode by pressing the SPACEBAR.

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

You may continue to add line segments as desired by clicking the S8 LEFT MOUSE or pressing the INSERT key. If you make a mistake, you can press DELETE to remove the last placed 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

Bus
Place Bus Entry

Place Bus Entry

Summary Place bus entry on the worksheet.

To Launch Alt P U - 

Purpose The Place Bus Entry command is used to place a graphical Bus Entry object on the current worksheet. A Bus Entry is used to connect wires to a bus line.

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

Procedure To place a Bus Entry:

1. Choose Bus Entry from the Place menu.

Press TAB to manually change the default values of the Bus line by typing new values directly into a dialog box. As you move the mouse or press the arrow keys, a highlighted Bus Entry segment will be displayed. You can change the Bus Entry angle by pressing the SPACEBAR.

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

You may continue to add bus entry objects as desired by moving and clicking the mouse.

3. To exit this command press ESC or RIGHT MOUSE.

See also

Bus Entry
Place Bus

Place Parts

Summary Place parts on the worksheet.

To Launch Alt P P - 

Purpose The Place Part command is used to retrieve a library part for placement on the current worksheet. During placement the part may be rotated or mirrored.

Comments The component library must be listed in the Library File List dialog box, before you can place it on the worksheet, use the Add/Remove Library command to add a library to the list. Refer to the user guide for detailed descriptions about Component and Part management.

Procedure To Place Parts on the current worksheet.

1. Choose Parts from the Place menu.

The Component Library Reference Name dialog box opens, displaying the name of the last placed Part.

2. Enter the Library Reference Name (74LS00, for example) into the dialog box and click OK or press ENTER.

3. Press ENTER or click OK to accept the default designator or type in a new designator then click OK or press ENTER. The Part will then be displayed in the workspace.

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 once.

5. The Component Library Reference Name dialog box will be displayed again, type in another Library Reference Name or press ESC or Cancel to exit this command.

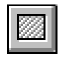
You can also place parts from the Browser by clicking the place button. The Browser allows browsing of component names, placement of parts onto the current worksheet, renaming of part designators and jumping to a part that has been placed on a worksheet in a project.

See also

Add/Remove Library
Place Part from Browser

Place Rectangles

Summary Place rectangles on the worksheet.

To Launch Alt P D R - 

Purpose The Place Rectangle command is used to place a graphical rectangle on the current schematic worksheet. Rectangles can be used for adding reference information to a worksheet, such as building graphical symbols, custom sheet borders, and title blocks.

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 current worksheet:

1. Choose Drawing Tools and then 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 pressing the 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.

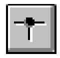
Place another rectangle or press ESC or RIGHT MOUSE to exit this command.

See also

Rectangle
Place Round Rectangle

Place Junctions

Summary Place a junction on the current worksheet.

To Launch Alt P J - 

Purpose Junctions are used to electrically connect intersecting wires on a sheet.

Comments Crossed or tangent wires in a schematic worksheet are not deemed to be connected unless a junction is placed where they intersect. Junctions are also placed where a wire crosses component pins.

When using the Place Junction command the default placement field values are determined by the last placed junction. Press TAB to change the default junction size and color during placement.

When the Auto Junction option (Options-Preferences command) is enabled, junctions are automatically added/removed from the sheet as wires and/or parts are placed and moved.

Procedure To place a junction on the current worksheet:

1. Choose Place and then Junction 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. You can tell when electrical hot spots are in contact because their shape changes into a large dot.

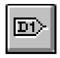
Place another Junction or press ESC or RIGHT MOUSE to exit this command.

See also

Junction

Place Port

Summary Place a port on the sheet.

To Launch Alt P R - 

Purpose Ports are used to indicate connections between a net on two individual sheets in both flat and hierarchical designs.

Comments All Ports with the same name, within one sheet, are considered to be electrically connected. Ports can be used within a worksheet or across multiple sheets within a project.

A Ports 'Style' is independent from its I/O Type. For example, a Port with the Style of 'Left & Right' does not necessarily mean the Port is Bi-directional.

When using the Place Port command the default placement field values are determined by the last placed port. Press TAB to change the default port attributes, style, I/O type, alignment, border color, fill color and text color during placement.

Procedure To place a Port on the schematic worksheet:

1. Choose Port from the Place menu.

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

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

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

3. Position the cursor to establish the desired length of the Port, then press ENTER or LEFT MOUSE.

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

See also

Port

Place Net Label

Summary Place a net label on the worksheet.

To Launch Alt P N - 

Purpose The Place Net Labels command is used to assign a wire and its entire physical net to a specific net name. Net labels are also used to connect signals (wires) together without actually physically connecting them. You can place labels horizontally or vertically on a worksheet.

Comments Within a sheet, all Net Labels with the same name are considered to be electrically connected. Net labels can also be made global -- applying to nets with the same name on all project sheets. This is done by changing the Net Identifiers Scope to Nets and Ports Global, under the File Create Netlist command.

When using the Place Net label command the default placement field values are determined by the last placed net label. Press TAB to change the default net label attributes, orientation, color and font style during placement.

Procedure To place a Net Label on the schematic worksheet.

1. Choose Net Label from the Place menu.

While moving the Net Label, 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 Net Label by typing new values directly into a dialog box.

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

You can tell when electrical hot spots are in contact because their shape changes into a large dot.

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

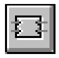
3. Place another Net Label or press ESC or RIGHT MOUSE to exit this command.

See also

Net Label

Place Sheet Symbol

Summary Place hierarchical sheet symbols on the worksheet.

To Launch Alt P S - 

Purpose The Place Sheet Symbol command is used to represent another schematic sheet in a hierarchical design. Sheet symbols include Sheet Entrys, which provide a connection point for signals between the parent and child sheets, similarly to the way that Ports provide connections between sheets in a flat-sheet design.

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

Procedure To place a Sheet Symbol onto the schematic worksheet.

1. Choose Sheet Symbol from the Place menu.

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

2. Position the cursor where one corner of the Sheet Symbol 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.

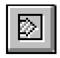
Place another Sheet Symbol or press ESC or RIGHT MOUSE to exit this command.

See also

Sheet Symbol

Add Sheet Entry

Summary Add net connections to sheet symbols.

To Launch Alt P E - 

Purpose The Add Sheet Entry command is used to direct signals to another sheet in a hierarchical design. There are four types of Sheet Entry symbols, Input, Output, Bi-directional and Unspecified.

Comments Sheet Entry symbols are similar to Ports in that they direct a signal to another sheet. Sheet Entry symbols provide a connection point for signals going into and out of Sheet Symbols.

When using the Place Add Sheet Entry command the default placement field values are determined by the last placed sheet entry. Press TAB to change the default sheet entry name, electrical type, style, border color, fill color, and text color during placement.

Procedure To place a Sheet Entry onto the schematic worksheet.

1. Choose Add Sheet Entry from the Place menu.

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

2. Position the cursor over a Sheet Symbol where the Sheet Entry is to be placed and press ENTER or LEFT MOUSE once.

If the system beeps you may not be exactly over a Sheet Symbol. Use the Zoom options (or press PGUP) to enlarge your view of the layout and try again.

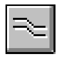
Place another Sheet Entry or press ESC or RIGHT MOUSE to exit this command.

See also

Sheet Entry

Place Wire

Summary Place electrical wires on the worksheet.

To Launch Alt P W - 

Purpose The Place Wire command is used to place electrical Wire connections on the current schematic worksheet. The wires represent electrical connections between Parts on the schematic sheets.

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

Procedure To place a wire on the current worksheet:

1. Choose Wire from the Place menu.

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

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

As you move the mouse or press the arrow keys, a highlighted Wire segment will be displayed. You can change the Wire placement mode by pressing the SPACEBAR.

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

You may continue to add Wire 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 placed line segment. You can also press ESC or RIGHT MOUSE to Cancel the current segment.

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

See also

Wire

Place Power Port

Summary Place power ports on the worksheet.

To Launch Alt P O - 

Purpose The Place Power Port command is used to place Power Ports on the schematic worksheet. Power Ports are used to tie signals and pins to a power or ground net, without having to place net labels .

Comments When using the Place Power Port command the default placement field values are determined by the last placed power port. Press TAB to change the default power port name, style, orientation and color during placement.

Procedure To place a Power on the schematic worksheet.

1. Choose Power Port from the Place menu.

While moving the Power Port, you can rotate it 90 degrees around the cursor 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 Power Port by typing new values directly into a dialog box.

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

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

3. Place another Power Port or press ESC or RIGHT MOUSE to exit this command.

See also

Power Port

Place No ERC

Summary Suppress no-connection error message for nets.

To Launch Alt P I N - 

Purpose No ERCs are special symbols that identify a pin on a part that is to be left unconnected. During electrical design rules checking, unconnected pins with No ERC symbols will be ignored.

Comments No ERC symbols must be placed on the end of pins in order for the No ERC to be effective.

When using the Place No ERC command the default placement field values are determined by the last placed No ERC symbol. Press TAB to change the default No ERC color during placement.

Procedure To place a No ERC symbol on the schematic worksheet.

1. Choose Directives and then No ERC from the Place menu.

Press TAB to manually change the default values of the No ERC symbol by typing new values directly into a dialog box.

2. Position the cursor where the No ERC symbol is to be placed and press ENTER or LEFT MOUSE once.

As you move the mouse or press the arrow keys, a highlighted No ERC symbol will be displayed.

3. Place another No ERC symbol or press ESC or RIGHT MOUSE to exit this command.

See also

No ERC

Place Probe

Summary Add net to list of nets to be traced by digital simulator.

To Launch Alt P I P - 

Purpose A Probe is a special marker which is placed on the worksheet to identify nodes for digital simulation.

Comments When using the Place Probe command the default placement field values are determined by the last placed probe. Press TAB to change the default probe attributes and color during placement.

Probes are used in OrCAD digital simulation tools, refer to your OrCAD documentation for further information about using simulation probes.

Procedure To place a Probe on the schematic worksheet.

1. Choose Directives and then Probe from the Place menu.

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

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

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

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

See also

Probe

Place Test Vector Index

Summary Mark nets to identify which simulation vector to use.

To Launch Alt P I T

Purpose Test Vectors are special symbols used to identify a node with a simulation test vector. The test vectors are referred to by a column number, which indicates the column of the test vector file to use when the simulation is run.

Comments When using the Place Test Vector Index command the default placement field values are determined by the last placed Test Vector Index. Press TAB to change the default Test Vector Index attributes and color during placement.

Test Vector Index symbols are used in OrCAD digital simulation tools, refer to your OrCAD documentation for further information about using simulation vectors.

Procedure To place a Text Vector Index on the schematic worksheet.

1. Choose Directives and then No ERC from the Place menu.

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

2. Position the cursor on the wire where the Test Vector Index is to be placed and press ENTER or LEFT MOUSE once.

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

3. Place another Test Vector Index or press ESC or RIGHT MOUSE to exit this command.

See also

Test Vector Index

Place Stimulus

Summary Add stimulus information to nets for digital simulation.

To Launch Alt P I S - 

Purpose A Stimulus is a special symbol which is used to identify a node to be stimulated when the digital simulation is run. The stimulus is an expression describing the pattern of states present in the circuit logic.

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

Stimulus symbols are used in OrCAD digital simulation tools, refer to your OrCAD documentation for further information about using simulation stimulus.

Procedure To place a Stimulus on the schematic worksheet.

1. Choose Directives and then Stimulus from the Place menu.

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

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

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

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

See also

Stimulus

Place PCB Layout

Summary Add directives to nets for PCB routing.

To Launch Alt P I P - 

Purpose PCB Layouts are special symbols that allow you to attach board layout information to a specific net. This information is saved into the netlist file which is later passed to Advanced PCB, OrCAD PCB or other compatible board layout systems.

Comments When using the Place PCB Layout command the default placement field values are determined by the last placed PCB Layout. Press TAB to change the default probe attributes and color during placement.

Procedure To place a PCB Layout symbol on the schematic worksheet.

1. Choose Directives and then PCB Layout from the Place menu.

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

2. Position the cursor on the wire where the PCB Layout symbol is to be placed and press ENTER or LEFT MOUSE once.

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

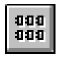
3. Place another PCB Layout symbol or press ESC or RIGHT MOUSE to exit this command.

See also

PCB Layout

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 net labels, annotations and part designators. Setting this to 1 will automatically increment. For example, an initial Net Label of Net1 will automatically increment to Net2, Net3, Net4 and so on, and a Net Label of 1Net will increment 2Net, 3Net, 4Net.

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 LEFT MOUSE or ENTER.

See also

Cut
Copy
Execute Paste Array
Place Array Information

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 (Options-Hot Keys command) 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 LEFT MOUSE or ENTER.

See also

Paste Array
Hot Keys

Search For Net

Summary Search for and jump to the closest point of a net.

Purpose The Search for Net process allows you to conveniently locate a specific net on the worksheet without having to zoom, pan or scroll through multiple screens.

Outcome Once the net name has been entered or selected from a list the cursor will jump to the nearest pin that belongs to the selected net.

Comments The Browser supports jumping directly to nets. This process allows users to jump to nets when the Browser is hidden. The Find Text command can also be used to search for net labels (or any object with text).

Procedure To search for a Net:

1. Assign the process Search For Net to any key combination (Options-Hot Keys command).

2. When activated, this process will open the Search For Net dialog box.

3. Type the net name in the Search For dialog box and click OK.

See also

Find Text

Summary Search for and jump to a text string on any sheet.

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 worksheet 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 on a sheet or across a multi-sheet project. Extends across all objects with visible text, including Part Fields and Text Frames.

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.

See also

Jump Origin

Summary Jump to the origin of the worksheet (lower left).

To Launch Alt E J O

Purpose The Jump Origin command is used to move the cursor to the lower left corner of the schematic worksheet. 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 schematics.

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

Procedure To move the cursor to the origin of the worksheet:

1. Choose Jump Origin from the Edit menu.

See also

Jump New Location

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 design and is particularly useful for large or complex schematics.

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 left hand side of the worksheet), the default is the current X position.

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

3. Click OK to jump to the specified location.

See also

Jump Origin

Jump Location Mark 1

Summary Jump to a location previously specified by setting Location Mark 1-10..

To Launch Alt E J 1

Purpose The Jump Location Mark command is used to move the cursor to a pre-defined location on the schematic worksheet. 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 schematics.

Outcome Once the Jump Location Mark command has been selected the cursor will jump to the location that was set for the Location Mark.

Comments The Location Mark is set by using the Location Mark commands in the Place menu. If the Location Mark has not been set the cursor will remain at the current location.

Procedure To move the cursor to the Location Mark:

1. Choose Jump Location Mark (1-10) from the Edit menu.

See also

Set Location Mark 1

Jump Location Mark 2

Summary Jump to a location previously specified by setting Location Mark 1-10..

To Launch Alt E J 2

Purpose The Jump Location Mark command is used to move the cursor to a pre-defined location on the schematic worksheet. 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 schematics.

Outcome Once the Jump Location Mark command has been selected the cursor will jump to the location that was set for the Location Mark.

Comments The Location Mark is set by using the Location Mark commands in the Place menu. If the Location Mark has not been set the cursor will remain at the current location.

Procedure To move the cursor to the Location Mark:

1. Choose Jump Location Mark (1-10) from the Edit menu.

See also

Set Location Mark 2

Jump Location Mark 3

Summary Jump to a location previously specified by setting Location Mark 1-10..

To Launch Alt E J 3

Purpose The Jump Location Mark command is used to move the cursor to a pre-defined location on the schematic worksheet. 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 schematics.

Outcome Once the Jump Location Mark command has been selected the cursor will jump to the location that was set for the Location Mark.

Comments The Location Mark is set by using the Location Mark commands in the Place menu. If the Location Mark has not been set the cursor will remain at the current location.

Procedure To move the cursor to the Location Mark:

1. Choose Jump Location Mark (1-10) from the Edit menu.

See also

Set Location Mark 3

Jump Location Mark 4

Summary Jump to a location previously specified by setting Location Mark 1-10..

To Launch Alt E J 4

Purpose The Jump Location Mark command is used to move the cursor to a pre-defined location on the schematic worksheet. 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 schematics.

Outcome Once the Jump Location Mark command has been selected the cursor will jump to the location that was set for the Location Mark.

Comments The Location Mark is set by using the Location Mark commands in the Place menu. If the Location Mark has not been set the cursor will remain at the current location.

Procedure To move the cursor to the Location Mark:

1. Choose Jump Location Mark (1-10) from the Edit menu.

See also

Set Location Mark 4

Jump Location Mark 5

Summary Jump to a location previously specified by setting Location Mark 1-10..

To Launch Alt E J 5

Purpose The Jump Location Mark command is used to move the cursor to a pre-defined location on the schematic worksheet. 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 schematics.

Outcome Once the Jump Location Mark command has been selected the cursor will jump to the location that was set for the Location Mark.

Comments The Location Mark is set by using the Location Mark commands in the Place menu. If the Location Mark has not been set the cursor will remain at the current location.

Procedure To move the cursor to the Location Mark:

1. Choose Jump Location Mark (1-10) from the Edit menu.

See also

Set Location Mark 5

Jump Location Mark 6

Summary Jump to a location previously specified by setting Location Mark 1-10..

To Launch Alt E J 6

Purpose The Jump Location Mark command is used to move the cursor to a pre-defined location on the schematic worksheet. 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 schematics.

Outcome Once the Jump Location Mark command has been selected the cursor will jump to the location that was set for the Location Mark.

Comments The Location Mark is set by using the Location Mark commands in the Place menu. If the Location Mark has not been set the cursor will remain at the current location.

Procedure To move the cursor to the Location Mark:

1. Choose Jump Location Mark (1-10) from the Edit menu.

See also

Set Location Mark 6

Jump Location Mark 7

Summary Jump to a location previously specified by setting Location Mark 1-10..

To Launch Alt E J 7

Purpose The Jump Location Mark command is used to move the cursor to a pre-defined location on the schematic worksheet. 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 schematics.

Outcome Once the Jump Location Mark command has been selected the cursor will jump to the location that was set for the Location Mark.

Comments The Location Mark is set by using the Location Mark commands in the Place menu. If the Location Mark has not been set the cursor will remain at the current location.

Procedure To move the cursor to the Location Mark:

1. Choose Jump Location Mark (1-10) from the Edit menu.

See also

Set Location Mark 7

Jump Location Mark 8

Summary Jump to a location previously specified by setting Location Mark 1-10..

To Launch Alt E J 8

Purpose The Jump Location Mark command is used to move the cursor to a pre-defined location on the schematic worksheet. 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 schematics.

Outcome Once the Jump Location Mark command has been selected the cursor will jump to the location that was set for the Location Mark.

Comments The Location Mark is set by using the Location Mark commands in the Place menu. If the Location Mark has not been set the cursor will remain at the current location.

Procedure To move the cursor to the Location Mark:

1. Choose Jump Location Mark (1-10) from the Edit menu.

See also

Set Location Mark 8

Jump Location Mark 9

Summary Jump to a location previously specified by setting Location Mark 1-10..

Purpose The Jump Location Mark command is used to move the cursor to a pre-defined location on the schematic worksheet. 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 schematics.

Outcome Once the Jump Location Mark command has been selected the cursor will jump to the location that was set for the Location Mark.

Comments The Location Mark is set by using the Location Mark commands in the Place menu. If the Location Mark has not been set the cursor will remain at the current location.

Procedure To move the cursor to the Location Mark:

1. Choose Jump Location Mark (1-10) from the Edit menu.

See also

Set Location Mark 9

Jump Location Mark 10

Summary Jump to a location previously specified by setting Location Mark 1-10..

Purpose The Jump Location Mark command is used to move the cursor to a pre-defined location on the schematic worksheet. 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 schematics.

Outcome Once the Jump Location Mark command has been selected the cursor will jump to the location that was set for the Location Mark.

Comments The Location Mark is set by using the Location Mark commands in the Place menu. If the Location Mark has not been set the cursor will remain at the current location.

Procedure To move the cursor to Location Mark:

1. Choose Jump Location Mark (1-10) from the Edit menu.

See also

Set Location Mark 10

Set Location Mark 1

Summary Set location mark (1-10) on the current worksheet.

To Launch Alt P L 1

Purpose The Set Location Mark command is used to tag or mark a specific location on the schematic worksheet. The Jump Location Mark command is used to move the cursor to this set location.

Outcome Once the Set Location Mark command has been selected you will be prompted to select a location to be stored for the Location Mark, move the cursor to the desired location and click OK or press ENTER.

Comments To move the cursor to this location you must use the Jump Location Mark command.

Procedure To define a new location for the Location Mark:

1. Choose Jump Location Mark from the Edit menu.

2. Position the cursor at the desired location and click LEFT MOUSE or press ENTER.

The location will be recorded for the Location Mark.

See also

Jump Location Mark 1

Set Location Mark 2

Summary Set location mark (1-10) on the current worksheet.

To Launch Alt P L 2

Purpose The Set Location Mark command is used to tag or mark a specific location on the schematic worksheet. The Jump Location Mark command is used to move the cursor to this set location.

Outcome Once the Set Location Mark command has been selected you will be prompted to select a location to be stored for the Location Mark, move the cursor to the desired location and click OK or press ENTER.

Comments To move the cursor to this location you must use the Jump Location Mark command.

Procedure To define a new location for the Location Mark:

1. Choose Jump Location Mark from the Edit menu.

2. Position the cursor at the desired location and click LEFT MOUSE or press ENTER.

The location will be recorded for the Location Mark.

See also

Jump Location Mark 2

Set Location Mark 3

Summary Set location mark (1-10) on the current worksheet.

To Launch Alt P L 3

Purpose The Set Location Mark command is used to tag or mark a specific location on the schematic worksheet. The Jump Location Mark command is used to move the cursor to this set location.

Outcome Once the Set Location Mark command has been selected you will be prompted to select a location to be stored for the Location Mark, move the cursor to the desired location and click OK or press ENTER.

Comments To move the cursor to this location you must use the Jump Location Mark command.

Procedure To define a new location for the Location Mark:

1. Choose Jump Location Mark from the Edit menu.

2. Position the cursor at the desired location and click LEFT MOUSE or press ENTER.

The location will be recorded for the Location Mark.

See also

Jump Location Mark 3

Set Location Mark 4

Summary Set location mark (1-10) on the current worksheet.

To Launch Alt P L 4

Purpose The Set Location Mark command is used to tag or mark a specific location on the schematic worksheet. The Jump Location Mark command is used to move the cursor to this set location.

Outcome Once the Set Location Mark command has been selected you will be prompted to select a location to be stored for the Location Mark, move the cursor to the desired location and click OK or press ENTER.

Comments To move the cursor to this location you must use the Jump Location Mark command.

Procedure To define a new location for the Location Mark:

1. Choose Jump Location Mark from the Edit menu.

2. Position the cursor at the desired location and click LEFT MOUSE or press ENTER.

The location will be recorded for the Location Mark.

See also

Jump Location Mark 4

Set Location Mark 5

Summary Set location mark (1-10) on the current worksheet.

To Launch Alt P L 5

Purpose The Set Location Mark command is used to tag or mark a specific location on the schematic worksheet. The Jump Location Mark command is used to move the cursor to this set location.

Outcome Once the Set Location Mark command has been selected you will be prompted to select a location to be stored for the Location Mark, move the cursor to the desired location and click OK or press ENTER.

Comments To move the cursor to this location you must use the Jump Location Mark command.

Procedure To define a new location for the Location Mark:

1. Choose Jump Location Mark from the Edit menu.

2. Position the cursor at the desired location and click LEFT MOUSE or press ENTER.

The location will be recorded for the Location Mark.

See also

[Jump Location Mark 5]

Set Location Mark 6

Summary Set location mark (1-10) on the current worksheet.

To Launch Alt P L 6

Purpose The Set Location Mark command is used to tag or mark a specific location on the schematic worksheet. The Jump Location Mark command is used to move the cursor to this set location.

Outcome Once the Set Location Mark command has been selected you will be prompted to select a location to be stored for the Location Mark, move the cursor to the desired location and click OK or press ENTER.

Comments To move the cursor to this location you must use the Jump Location Mark command.

Procedure To define a new location for the Location Mark:

1. Choose Jump Location Mark from the Edit menu.

2. Position the cursor at the desired location and click LEFT MOUSE or press ENTER.

The location will be recorded for the Location Mark.

See also

Jump Location Mark 6

Set Location Mark 7

Summary Set location mark (1-10) on the current worksheet.

To Launch Alt P L 7

Purpose The Set Location Mark command is used to tag or mark a specific location on the schematic worksheet. The Jump Location Mark command is used to move the cursor to this set location.

Outcome Once the Set Location Mark command has been selected you will be prompted to select a location to be stored for the Location Mark, move the cursor to the desired location and click OK or press ENTER.

Comments To move the cursor to this location you must use the Jump Location Mark command.

Procedure To define a new location for the Location Mark:

1. Choose Jump Location Mark from the Edit menu.

2. Position the cursor at the desired location and click LEFT MOUSE or press ENTER.

The location will be recorded for the Location Mark.

See also

Jump Location Mark 7

Set Location Mark 8

Summary Set location mark (1-10) on the current worksheet.

To Launch Alt P L 8

Purpose The Set Location Mark command is used to tag or mark a specific location on the schematic worksheet. The Jump Location Mark command is used to move the cursor to this set location.

Outcome Once the Set Location Mark command has been selected you will be prompted to select a location to be stored for the Location Mark, move the cursor to the desired location and click OK or press ENTER.

Comments To move the cursor to this location you must use the Jump Location Mark command.

Procedure To define a new location for the Location Mark:

1. Choose Jump Location Mark from the Edit menu.

2. Position the cursor at the desired location and click LEFT MOUSE or press ENTER.

The location will be recorded for the Location Mark.

See also

Jump Location Mark 8

Set Location Mark 9

Summary Set location mark (1-10) on the current worksheet.

To Launch Alt P L 9

Purpose The Set Location Mark command is used to tag or mark a specific location on the schematic worksheet. The Jump Location Mark command is used to move the cursor to this set location.

Outcome Once the Set Location Mark command has been selected you will be prompted to select a location to be stored for the Location Mark, move the cursor to the desired location and click OK or press ENTER.

Comments To move the cursor to this location you must use the Jump Location Mark command.

Procedure To define a new location for the Location Mark:

1. Choose Jump Location Mark from the Edit menu.

2. Position the cursor at the desired location and click LEFT MOUSE or press ENTER.

The location will be recorded for the Location Mark.

See also

Jump Location Mark 9

Set Location Mark 10

Summary Set location mark (1-10) on the current worksheet.

To Launch Alt P L 0

Purpose The Set Location Mark command is used to tag or mark a specific location on the schematic worksheet. The Jump Location Mark command is used to move the cursor to this set location.

Outcome Once the Set Location Mark command has been selected you will be prompted to select a location to be stored for the Location Mark, move the cursor to the desired location and click OK or press ENTER.

Comments To move the cursor to this location you must use the Jump Location Mark command.

Procedure To define a new location for the Location Mark:

1. Choose Jump Location Mark from the Edit menu.

2. Position the cursor at the desired location and click LEFT MOUSE or press ENTER.

The location will be recorded for the Location Mark.

See also

Jump Location Mark 10

Zoom Window

Summary Select a rectangular area of the sheet 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 sheet 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 to "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 to "Select Second Corner."

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

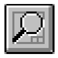
4. Press ENTER or LEFT MOUSE to complete the zoom window.

See also

Zoom Window
Zoom In
Zoom Out

Zoom Sheet

Summary Show entire sheet.

To Launch Alt Z S - 

Purpose The Zoom Sheet command is used to view the entire area of the current worksheet.

Procedure To view all of the current worksheet:

1. Choose Sheet from the Zoom menu.

See also

Zoom All

Zoom All

Summary Fit all objects on the current sheet 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 worksheet 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 worksheet into the current document window:

1. Choose All from the Zoom menu.

See also

Zoom Sheet

Zoom In

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

To Launch PgUp - Alt Z I - 

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

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 worksheet:

1. Press PGUP or z, i.

See also

[Zoom Out]
[Zoom All]

Zoom Out

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

To Launch PgDn - Alt Z O - 

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

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 worksheet:

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 worksheet 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 Screen Redraw command is used to update or refresh all objects on the current worksheet.

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

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 worksheet to a factor of 4 times the normal scale. This is the highest zoom magnification.

Outcome The view of the current worksheet 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 worksheet to 400%:

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

2. Choose 400% from the Zoom menu or shortcut: 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 worksheet to a factor of 2 times the normal scale.

Outcome The view of the current worksheet 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 worksheet to 200%:

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

2. Choose 200% from the Zoom menu or shortcut: 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 worksheet to its normal scale.

Outcome The view of the current worksheet 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 worksheet to 100%:

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

2. Choose 100% from the Zoom menu or shortcut 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 worksheet to a factor of half the normal scale.

Outcome The view of the current worksheet 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 worksheet to 50%:

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

2. Choose 50% from the Zoom menu or shortcut z, 5.

See also

Zoom All

Information Selected Pins

Summary List all selected pins.

To Launch Alt I E

Purpose The Information Selected Pins command is used to display a list of the part and pin designators currently selected on the worksheet. This command is useful to identify that your nets contain all the necessary pin information.

Comments This command provides a list of selected pins, plus sheet entries and ports. If you click on any listed items and press OK, the workspace will zoom to display that object.

Procedure To get information about the selected pins on the current worksheet:

1. Select pins by using the Select Net or Select Connection commands.

2. Choose Selected Pins from the Info menu.

A dialog box will appear with a list of the selected pins on the current worksheet.

See also

Select Net
Select Connection

Sheet Options

Summary Setup options for current worksheet.

To Launch Alt O S

Purpose The Options Sheet command is used to define various sheet settings, such as sheet styles, size, orientation, background and border colors and other options that apply to the worksheet.

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

Border Style - Advanced Schematic provides two default borders. You can use either the Standard or the ANSI border that uses the ANSI convention to provide a reference grid system.

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

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

Custom X - The worksheet size along the x-axis. The maximum X size is 65 inches, (6500 units).

Custom Y - The worksheet size along the y-axis. The maximum Y size is 65 inches, (6500 units).

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

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 Advanced Schematic's drawing tools.

Title Block - Advanced Schematic provides a default title block. You can use either the Standard title block or the ANSI standard title block. Some of the information in the title block is provided automatically, e.g. the sheet size, file name and creation date. Other information can be added using Annotations or Text Frames. You can hide the supplied title block by disabling this option and then use Advanced Schematics' Drawing tools to create custom title blocks.

Border Color - The surrounding border and title block of your schematic worksheet can be assigned a default color. To assign a new color to the sheet border and title block, click in the color box to open the Color Selector dialog box. Advanced Schematic will display all the available colors that your graphics card and monitor supports. The border color only applies to Advanced Schematics' default border and default title block. Customized borders and title blocks will not be affected by this option.

Color - A worksheet can be assigned a color. To assign a new color to the worksheet, click in the color box to open the Color Selector dialog box. Advanced Schematic will display all the available colors that your graphics card and monitor supports.

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 sheet settings;

1. Choose Sheet from the Options menu

2. Type in new sheet 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 schematic worksheet.

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 schematic worksheet 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 Sheet dialog box.

Procedure To turn the visible grid on or off:

1. Choose Visible Grid from the Options menu.

See also

Preferences
Sheet 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 worksheet 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 schematic worksheet.

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 UP ARROW on the keyboard while holding down SHIFT.

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 schematic worksheet.

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 DOWN ARROW on the keyboard while holding down SHIFT.

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 schematic worksheet.

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 LEFT ARROW on the keyboard while holding down SHIFT.

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 schematic worksheet.

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 RIGHT ARROW on the keyboard while holding down SHIFT.

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 schematic worksheet.

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 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 schematic worksheet.

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 DOWN ARROW on the keyboard.

See also

<u>**Shift Cursor Down**</u>

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 schematic worksheet.

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 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 schematic worksheet.

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 snap grid point:

1. Press RIGHT ARROW on the keyboard.

See also

Shift Cursor Right

Make Project Library

Summary Make a library of parts contained in the current project.

To Launch Alt L M

Purpose The Make Project Library command is used to automatically build a component library of all the parts that have been placed in a project.

Outcome The library file will be saved to the new file name, format and directory that you specified.

Comments The Make Project Library command can be used to generate an exact working library of your finished project. The library could accompany your schematic project files when you send them to other engineers, instead of sending your original libraries.

Procedure To make a component library of parts:

1. Choose Make Project Library from the Library 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 Format window.

3. Click OK to save the file.

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

Update Parts

Summary Update all parts from library information.

To Launch Alt L U

Purpose The Update Part command is used to update all parts in every single sheet that is open, from libraries listed in the Change Library File List dialog box.

Outcome Any changes that have been made to parts in the Library Editor will be reflected in those parts in the current project.

Comments Generates a text report listing any parts that have been updated.

Procedure To update the parts on the current project:

1. Choose Update Parts from the Library menu.

Netlist Compare

Summary Compare two Protel format netlists and generate a report.

To Launch Alt F R N

Purpose To generate a text report listing the differences between two Protel format netlists. This is used to compare the differences between two revisions of a project, or to compare the difference between a schematic netlist and the netlist of the completed board generated by the PCB application.

Outcome A text report is generated, listing differences between two netlist files.

Comments Works with Protel, Protel 2 and Tango format netlists. This report does not report component differences.

Procedure Choose File Reports Netlist Compare. You will be prompted to load two netlist files. Type in the name for the first netlist or search the available directories for the desired .NET file. Then type or choose the second netlist and click OK.

A report file is generated and opened, using the Notepad utility. This text file lists all matched nets (by name) and reports:

Total Matched Nets

Total Partially Matched Nets

Total Extra Nets in the first netlist

Total Extra Nets in the second netlist

Total Nets in the first netlist

Total Nets in in the second netlist

Create Sheet Symbol From Sheet

Summary Create a sheet symbol that represents the current sheet.

To Launch Alt F H Y

Purpose Used in hierarchical designs, to speed creation of sheet symbols for a "child" sheet. After creating a sheet, this command will automatically generate a sheet symbol, labeled with its file name, including sheet entries for each port in the sheet. Sheet entry electrical characteristics and and styles complement the ports in the original sheet.

Comments For each Port on the sheet, an a sheet entry will be created on the sheet symbol. If the Port is input, then the sheet entry will be output etc.

Procedure Go to the sheet where the new sheet symbol will be placed. Run this process. A list of all sheets will be displayed. Choose the sheet and a sheet symbol representing that sheet will be created in the current sheet. The name and file name of the sheet symbol will reflect the chosen sheet.

When this process is run, the user is presented the option to "Reverse Input/Output Directions." This option changes the sheet entry electrical types to be opposite the ports that they represent in the sheet symbol.

See also

Create Sheet From Sheet Symbol
Hierarchical and multi sheet support

Create Sheet From Sheet Symbol

Summary Creates a new sheet (File New command) and adds ports for sheet entries on the sheet symbol.

To Launch Alt F H S

Purpose Used in hierarchical design. After creating a sheet symbol (if following a top down design methodology), automatically creates a new sheet with ports for all sheet entries present in the sheet symbol. Ports electrical characteristics and styles complement the sheet entries in the original sheet symbol.

Comments Use the Save As command after using this process. Otherwise, the new sheet will not be saved and hence there will be no file to represent it. Warning: if you crate a sheet for a sheet symbol and a sheet of that name already exists in the same directory, the previous sheet will be over-written when the new sheet is saved.

Procedure Place a Sheet Symbol, add sheet Entries, setting their electrical characteristics. Run this process. A new empty sheet will be created with Ports placed for each sheet entry. The file name of the sheet will reflect the sheet symbol filename.

When this process is run, the user is presented the option to "Reverse Input/Output Directions." This option changes the new sheet's port electrical types to be opposite the sheet entries that they represent.

See also

Create Sheet Symbol From Sheet
Hierarchical and multi sheet support

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 Run Options

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 Run Options

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. Tools menu commands provide convenient ways 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 Run Options

Run Picture Editor

Summary Run the user-specified paint/draw program (default is PBRUSH.EXE).

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 Run Options

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 Run Options

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 Run Options

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 Run Options

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 Run Options

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 Run Options

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 Run Options

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 Run Options

Setup Run Options

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. Tools menu commands provide convenient ways to launch frequently used applications and utilities.

Outcome Opens the Setup Run Options dialog box.

Comments **Note:**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.

The Analog Simulator, Digital Simulator, Mixed Signal Simulator or the PLD/FPGA Compiler options specified in this dialog box can be launched automatically when generating a netlist (specified in the File-Netlist Generate dialog box).

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 Library Editor - Editor for schematic library files. Default is LIBEDIT.EXE.

PCB Layout - Application for Printed Circuit Board layout files. Default is PFW.EXE.

PLD/FPGA Compiler - Compiler for PLD/FPGA files. Default is PLD.EXE.

Analog Simulator - Application for Analog Simulator. Default is ANASIM.EXE.

Digital Simulator - Application for Digital Simulator. Default is DIGSIM.EXE.

Mixed Signal Simulator - Application for Mixed Signal Simulator. Default is MIXSIM.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

User Command Information
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 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. Tools menu commands provide convenient ways 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.

Note: 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.

See also

Setup Run Options

Help DOS Schematic

Summary Command Cross Reference for Protel DOS Schematic users.

To Launch H D

Purpose The Help DOS Schematic menu option displays the command cross-reference for Protel DOS Schematic users.

Outcome Once the Help DOS Schematic command is selected the Advanced Schematic on-line help system is loaded and displays a cross-reference of command topics. Refer to the User Guide and reference supplement for information about Protel DOS Schematic compatibility.

Procedure To display the Protel DOS Schematic command cross-reference:

1. Choose DOS Schematic from the Help menu.

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

See also

Schematic 3 commands
Schematic 3 libraries
Block, highlighting, etc
Utilities

Jump To Next Error

Summary Jump to the next Error Marker in the Project.

To Launch Alt E J E

Purpose Finding the next error in any sheet of the current project (starting at the current sheet).

Outcome Once the Jump to Error Marker command has been selected the cursor will jump to the next Error Marker in the Project.

Comments After running the Electrical Rule Checker use this command to jump to error marker which have been placed on the sheet. After correcting the error, delete the error marker or run the Electrical Rule Checker again. Place down a No ERC marker if you do not want this error to be reported.

Procedure To find the next error in the Project:

1. Choose Jump from the Edit menu and then Jump to Error Marker command.

See also

No ERC
Electrical Rules Check

Cross Probe Part on PCB

Summary Select part to cross probe to PCB.

To Launch Alt E O

Purpose Cross probing parts allows user to jump directly to a specified part (component footprint) in Protel for Windows PCB applications.

Outcome When the Cross Probe Part command is chosen, the PCB layout becomes the focus window and the selected component in the PCB layout is displayed in the center of the window.

Comments Implemented in Protel for Windows Advanced PCB version 1.5 or later.

Procedure To cross probe a part in the PCB layout:

1. Open the Protel for Windows PCB file that corresponds to the active schematic sheet file;

2. Choose Cross Probe Part from the Edit menu. You will be prompted to select a part;

3. Position the cursor directly over the part you wish to cross probe and click LEFT MOUSE.

The focus will move to the PCB window and the corresponding part will displayed in the center of the window.

See also

Cross Probe Net On PCB
Cross Probe Pin on PCB

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 Run Options

Popup Move Menu

Summary Popup the 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

Drag Single Item

Summary Focus on and move a single object (and drag connected wires).

Purpose Allows the user to move an item using focus, rather than selection, and drag electrically connected items.

Outcome As you move the item, any wires that are connected will drag to maintain the connection.

Comments While dragging wires the wire placement mode can be changed interactively by using the SPACEBAR. Objects can also be restricted to only move along the x-axis by holding down the SHIFT+CTRL keys, or along the y-axis by holding down the SHIFT key. To rotate the object press the CTRL+SPACEBAR keys.

Procedure 1. Click LEFT MOUSE on an item to place it in focus.

You can tell an item is in focus, because it is displayed using a dashed line and its graphical editing handles are visible.

2. Hold CTRL and click LEFT MOUSE again.

This command can also be assigned to a hot key by using the Hot Keys command from the Options menu.

See also

Drag
Drag Selection

Drag

Summary Select and move objects together with connected wires/buses.

To Launch Alt E M D

Outcome As you move the item, any wires or busses that are connected will drag to maintain the connection.

Comments While dragging wires the wire placement mode can be changed interactively by using the SPACEBAR. Objects can also be restricted to only move along the x-axis by holding down the SHIFT+CTRL keys, or along the y-axis by holding down the SHIFT key. To rotate the object press the CTRL+SPACEBAR keys.

Procedure 1. Choose Move Drag from the Edit menu;

The prompt "Choose Object to Drag" is displayed on the Status Line.

2. Position the cursor over the object to be moved and click LEFT MOUSE to move the object;

3. When the move is completed, click LEFT MOUSE again;

Note that the prompt "Choose Object to Drag" is still displayed on the Status Line.

4. Choose another item to move or press RIGHT MOUSE or ESC to exit this option.

See also

Drag Single Item
Drag Selection

Cross Probe Net On PCB

Summary Select power object / Net Label to cross probe to PCB.

To Launch Alt E B

Purpose Cross probing nets allows user to jump directly to a specified highlighted net in Protel for Windows PCB applications.

Outcome When the Cross Probe Net command is chosen, the PCB layout becomes the focus window and the selected net in the PCB layout is highlighted.

Comments Implemented in Protel for Windows Advanced PCB version 1.5 or later.

Procedure To cross probe a net in the PCB layout:

1. Open the Protel for Windows PCB file that corresponds to the active schematic sheet file;

2. Chosse Cross Probe Net from the Edit menu. You will be prompted to select the net label or power port to cross probe;

3. Position the cursor directly over the power port or net label you wish to cross probe and click LEFT MOUSE.

The focus will move to the PCB window and the corresponding net will be highlighted.

See also

Cross Probe Part on PCB
Cross Probe Pin on PCB

Drag Selection

Summary Move selected object(s) and drag connected objects.

To Launch Alt E M R

Purpose The Move Drag Selection command is used to reposition a selection containing one or more objects as a single entity, and to drag any electrically connected objects.

Outcome When selected objects are moved, the electrically connected wires and/or buses stretch to move with the selection.

Comments While dragging wires the wire placement mode can be changed interactively by using the SPACEBAR. Objects can also be restricted to only move along the x-axis by holding down the SHIFT+CTRL keys, or along the y-axis by holding down the SHIFT key. To rotate the object press the CTRL+SPACEBAR keys.

Procedure To move selected objects and drag connected objects in an open worksheet:

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 + Click LEFT MOUSE to add objects to the current selection or to de-select any selected objects.

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

3. You will be prompted to move the selected objects, click ENTER or LEFT MOUSE anywhere on the worksheet.

If the Object's Electrical Hot Spot option is enabled in the Preferences dialog box the cursor will jump to the Electrical Hot Spot of the closest object, otherwise the cursor will remain relative to the selected objects and the original chosen cursor position. 4. Position the selection in the worksheet and click ENTER or LEFT MOUSE.

See also

[Drag]
[Move Selection]

Help

Summary Information on new features.

To Launch H N

Purpose Contains information on new features that have been added to Advanced Schematic since publication of the i User Guide and reference manuals.

Outcome Once the Help New Features command is selected the Advanced Schematic on-line help system is loaded and displays a list of new features.

Procedure To display information about new features:

1. Choose New features from the Help menu.

See also

New Features for Release 2.00

File Close Project

Summary Close all open schematic windows.

Purpose The Close Project command is used to close all document windows and icons in the current project.

Outcome All sheets in the current (active) project are closed.

Procedure To close all sheets in the current (active) project:

1. Choose Close Project from the File menu;

You will be prompted "Changed. Save before Close?"

2. Click OK to close all sheets in the current project.

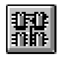
Sheets outside the project will remain opened.

See also

Window Close All

Run PCB Editor

Summary Switch to the PCB Layout Editor.

To Launch T P - 

Purpose This command is used to launch or swithch to the PCB Editor application directly from Advanced Schematic. Tools menu commands provide convenient ways to launch frequently used applications and utilities.

Outcome Launches the specified PCB Layout Editor.

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

If the PCB Layout Editor has been previously loaded then the focus will switch to the PCB application window, otherwise the PCB Editor will be launched, automatically loading previously opened PCB files.

Procedure Choose the PCB Layout Editor command from the Tools menu.

See also

Setup Run Options

Edit Object From Browser

Summary Changes the attributes associated with an object from the Browser.

Purpose The Edit Object From Browser command is used to change the current object from the Browser. The command is represented by the EDIT button on the Browser.

Outcome The object's attributes will be updated, all text and graphical changes such as object colors will be updated on the sheet.

Procedure To edit an object's attributes, first make sure that the object you wish to edit is highlighted in the Browser.

1. Press the Edit button on the Browser.

2. Type the new attributes into the dialog box and press ENTER or click OK.

See also

Part

View Read Only Part Fields

Summary View read-only fields for a part..

Purpose The View Read Only Part Fields command is used to view the read-only fields of a part on the wroksheet. The command is represented by the VIEW LIBRARY FIELDS button in the Part dialog box.

Comments The fields in the Read Only Part Fields dialog box cannot be changed.

Procedure To view the read-only fields of a part;

1. Edit a Part

2. Press the Global button in the Part dialog box and Choose the View Library Fields Button.

3. The Read Only Part Fields dialog box opens displaying field information for the Library Reference, Designator, Part Type, Part Description and Library Part Field 1 thru to Library Part Field 8.

This command can also be assigned to a hot key by using the Hot Keys command from the Options menu. To open the Read Only Part Fields dialog box with a hot key, make sure the object is focused and then activate the shortcut key.

See also

Part

Toggle Electrical Grid

Summary Turn the electrical grid on or off.

To Launch Alt O E

Purpose The Toggle Electrical Grid command is used to turn the cursor Electrical grid on or off. The Electrical grid defines an array of points in the workspace that restricts cursor movement and the placement of objects. Turning the Electrical grid off allows you to place objects at any location on the worksheet without any restrictions.

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

Procedure To turn the electrical grid on or off:

1. Choose Electrical Grid from the Options menu.

See also

Sheet Options

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 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
Find and Replace Text

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 on a sheet or across a multi-sheet project. Extends across all objects with visible text, including Part Fields and Text Frames.

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, Restrict To Net Identifiers (net labels, power ports, ports and sheet

entries).

6. Click OK to perform the text replacement.

See also

- [Find Text]
- [Find Next Text]

Update Current Template

Summary Update the current sheet from its template file..

To Launch Alt O D

Purpose The Update Current Template command is used to update the current sheets template information from its template file.

Outcome Any changes that have been made to the current sheet's template file will be reflected in the current sheet.

Comments The current contents of the sheets template will be deleted and new contents will be loaded. If the template file no longer exists in the original template path then the system path is checked if the template filename still cannot be found then there would be no action taken. Settings in the Options Sheet dialog box will be changed along with the Organization Name and Address information that is contained in the Document Info dialog box. Sheet Number, Sheet Total, Document Title, Document Number and Document Revision fields are not changed.

Procedure To update the current sheet's template

1. Choose Update Current Template from the Options menu.

2. A message box prompting to update template for all currently open sheets will be displayed. Choose YES to update the templates of all the sheets that are currently open, Choose NO to update only this sheet or CANCEL to exit from this command without any changes.

See also

Set Template File Name
Remove Template
Preferences

Convert Complex To Simple Hierarchy

Summary Convert Complex Hierachical Design To Simple Hierarchical Design.

To Launch Alt F H X

Purpose A simple hierachical design is one where each sheet is only used once in the design. A Complex hierarchy is one where a sheet can be used more than once. This makes the design much simpler, but when transfering to a physical design, such as a PCB, each part and net must be unique.

This command will convert a complex hierarchy to a simple hierarchy by duplicating the sheets that are used more than once.

Outcome New sheet windows will be created for the duplicated sheets

Procedure

1. Load the whole project to be converted

2. Set the focus on any window in the project to be converted

3. Choose Hierarchy from the File menu and then choose Complex To Simple.

Cross Probe Pin on PCB

Summary Select pin to cross probe to PCB.

To Launch Alt E P

Purpose Cross probing parts allows user to jump directly to a specified pin in Protel for Windows PCB applications.

Outcome When the Cross Probe Pin command is chosen, the PCB layout becomes the focus window and the selected pin in the PCB layout is displayed in the center of the window.

Comments Implemented in Protel for Windows Advanced PCB version 2.0 or later

Procedure To cross probe a pin in the PCB layout:

1. Open the Protel for Windows PCB file that corresponds to the active schematic sheet file;

2. Chosse Cross Probe Pin from the Edit menu. You will be prompted to select a pin;

3. Position the cursor directly over the pin you wish to cross probe and click LEFT MOUSE.

The focus will move to the PCB window and the corresponding pin will displayed in the center of the window.

See also

Cross Probe Net On PCB
Cross Probe Part on PCB

Change Object Graphically or Move

Summary Select and change objects on schematic sheet.

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.

While moving wires or busses pressing INSERT or DELETE) will add or remove a vertex/break in the wire or bus at the current cursor location.

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.

While editing or moving an object you can press the F1 key at anytime to open the Graphical Editing Hotkey List dialog box. This dialog box lists all the hot key commands that can be used while editing or moving the object.

See also

Change

Run PLD/FPGA Compile

Summary	Launch the PLD Compiler spcified in the Tools-Options Dialog.
To Launch	T F
Purpose	This command is used to launch the PLD/FPGA Compiler directly from Advanced Schematic. Tools menu commands provide convenient ways to launch frequently used applications and utilities.
Outcome	Launches the specified PCB Layout Editor.
Comments	User can specify the PLD/FPGA Compiler application under the Tools-Setup command.
Procedure	Choose the PCB Layout Editor command from the Tools menu.

See also

Setup Run Options

Popup Tools Menu

Summary Popup the Tools menu.

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

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

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

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

See also

Hot Keys
Setup Run Options

Run Digital Simulator

Summary Launch the Digital Simulator spcified in the Tools-Options Dialog.

To Launch T D

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

Outcome Launches the specified Digital Simulator.

Comments User can specify the Digital Simulator application under the Tools-Setup command.

Procedure Choose the Digital Simulator command from the Tools menu.

See also

Setup Run Options

Run Analog Simulator

Summary Launch the Analog Simulator spcified in the Tools-Options Dialog.

To Launch T A

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

Outcome Launches the specified Analog Simulator.

Comments User can specify the Analog Simulator application under the Tools-Setup command.

Procedure Choose the Analog Simulator command from the Tools menu.

See also

Setup Run Options

Run Mixed Signal Simulator

Summary Launch the Mixed-Signal Simulator spcified in the Tools-Options Dialog.

To Launch T X

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

Outcome Launches the specified Mixed Signal Simulator.

Comments User can specify the Mixed Signal Simulator application under the Tools-Setup command.

Procedure Choose the Mixed Signal Simulator command from the Tools menu.

See also

Setup Run Options

Set Template File Name

Summary Change the current template to a different template file.

To Launch Alt O N

Purpose The Set Template File Name command is used to change the current template of a sheet to a new template file name. A template is a special graphical entity that holds user-defined sheet size, border and title block descriptions. Users can create custom sheet templates and apply these templates to schematic sheets at any time, replacing the system's standard sheet definitions. They can be customized for specific organizations or purposes.

Outcome Any changes that have been made to the current sheet's template file will be reflected in the current sheet.

Comments The default file extension is .DOT. Sheet templates can be created from any schematic sheet at any time. The current contents of the sheets template will be removed and changed to the contents of the specified template file. Settings in the Options Sheet dialog box will be changed along with the Organization Name and Address information that is contained in the Document Info dialog box. Sheet Number, Sheet Total, Document Title, Document Number and Document Revision fields are not changed.

Procedure To change the current template.

1. Choose Set Template File Name 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

Remove Template
Update Current Template
Preferences

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 line.

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 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 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 This command is 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 or 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 sheet. 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.

Remove Template

Summary Remove any template information from the current sheet.

To Launch Alt O L

Purpose The Remove Template command is used to remove template information from the current sheet.

Procedure To remove graphical template information from the current sheet.

1. Choose Remove Template from the Options menu.

2. A message box prompting to remove templates from all currently open sheets will be displayed. Choose YES to remove the templates of all the sheets that are currently open, Choose NO to remove the template for the active sheet only or CANCEL to exit from this command without any changes.

See also

Set Template File Name
Update Current Template
Preferences

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

Align Objects
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

Find Component in Library

Summary Search for a component in a specified path..

To Launch Alt L F

Purpose The Find Component in Library command is used to search for a component that is in a specified drive/path, in the current listed libraries, or on all drives across a network. The component can be found by library reference name and/or by component description.

Outcome As components are found that match the search criteria, the component libraries are listed in the Found Libraries list box.

Comments To display the found components click on the library name in the Found Libraries list box, all the found component instances in that library will be listed in the Components list box. Clicking on the library name will display the library description and file details at the bottom of this dialog box, clicking on the component name will display the component description. The library and component descriptions are specified in the Library Editor. Press the Add To Library List button to automatically add the library to the Library File List.

Note: This command only searches for components in Advanced Schematic binary libraries. The time to search for a component by Description is considerably longer than searching by Library Reference.

Procedure To find a component in an Advanced Schematic binary library:

1. Choose Find Component from the Library menu.

2. The Find Component dialog box will open.

3. Type in the library reference name and/or component description into the Find Component fields. You can use wildcards such as * or ? to define the search criteria, for example, you can type in *74LS* to find all the components that have "74LS" in their name.

4. Specify the search criteria. The search scope can restrict the find to a specified drive/path, to the current libraries listed in the Library Add/Remove dialog box, or can search all the drives across a network or multiple networks.

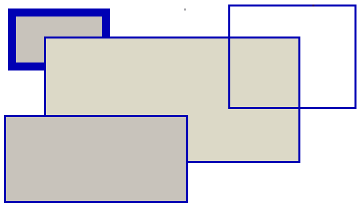
5. Press the Find button to start the search. You can abort the search at anytime by pressing the Stop button.

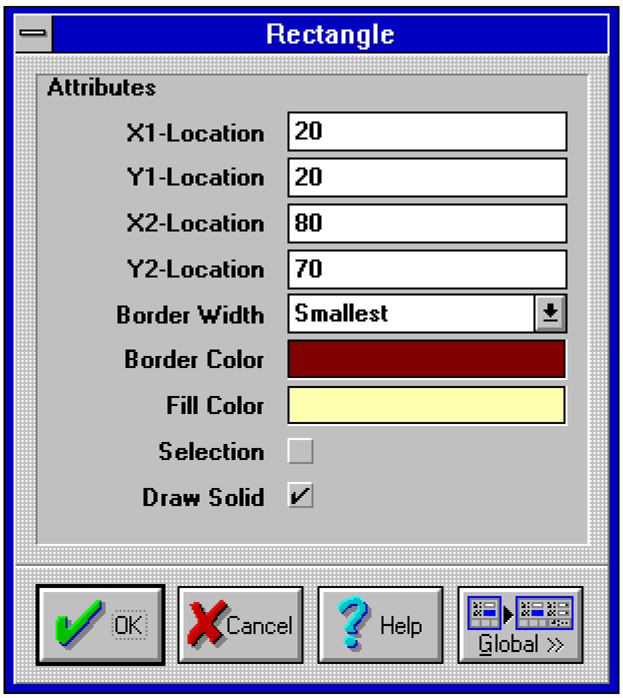
You can also activate the Find Component dialog box from the Browser by clicking the Find button.

Data Primitives

Rectangle

Overview Rectangles are filled or unfilled graphic elements placed on sheets, used for adding reference information to a worksheet, such as building graphical symbols, custom sheet borders, title blocks, etc..





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. Advanced Schematic 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. Advanced Schematic 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 Advanced Schematic 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.

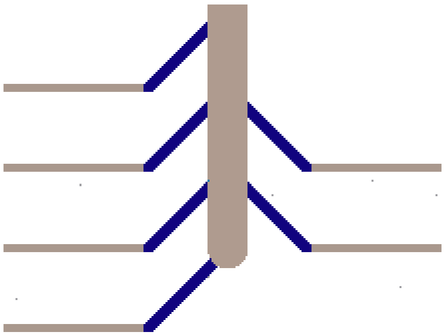
Comments Rectangles are graphical objects only, they have no logical or physical connectivity features.

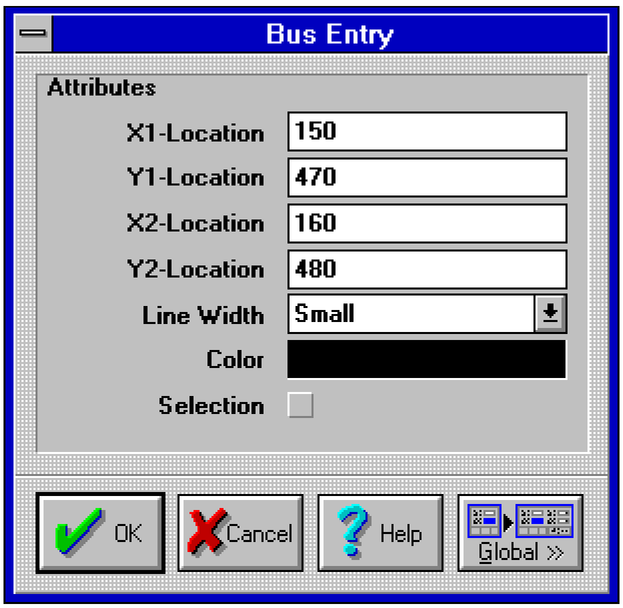
See also

Place Rectangles
Round Rectangle

Bus Entry

Overview Bus Entry objects are special graphical objects that are used to connect wires to a bus line. Using wires to connect to a bus can be limiting, especially when connecting different signals to the opposite sides of the bus line, thus causing an unwanted connection.





Fields

X1-Location The starting reference point coordinate of the bus entry, along the x-axis.

Y1-Location The starting reference point coordinate of the bus entry, along the y-axis.

X2-Location The ending reference point coordinate of the bus entry, along the x-axis.

Y2-Location The ending reference point coordinate of the bus entry, along the y-axis.

Line Width Specifies the line thickness of individual bus entry segments. There are four selectable line widths; smallest, small, medium and large.

Color Bus entry symbols can be assigned a color. To assign a new color to the bus entry segment, click in the color box to open the Color Selector dialog box. Advanced Schematic will display all the available colors that your graphics card and monitor supports.

Selection Toggle the selection state of bus entry symbols by turning this option on or off. If this option is on, the bus entry 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.

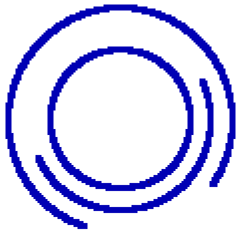
See also

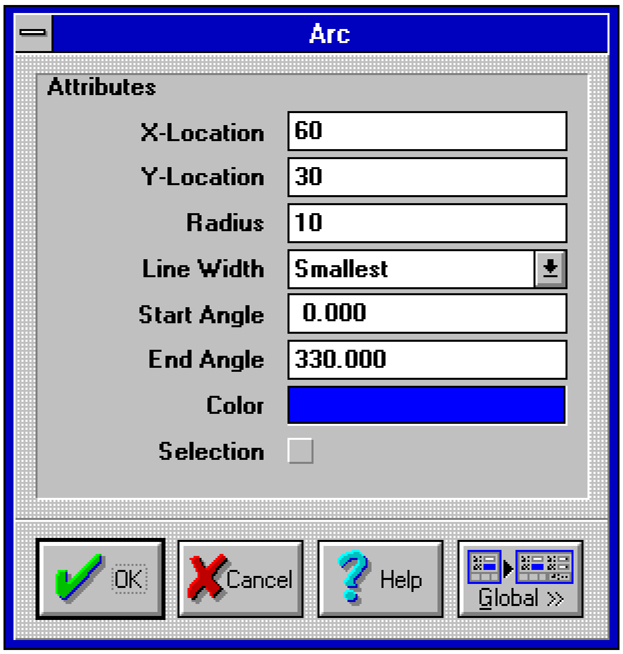
Place Bus Entry

Bus

Arc

Overview Arcs are circular line segments, used for adding reference information to a sheet, such as building graphical symbols, custom sheet borders, or title blocks.





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 Advanced Schematic 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 Advanced Schematic 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 Advanced Schematic 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. Advanced Schematic 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.

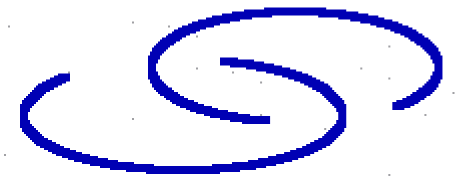
Comments Arcs have no logical or physical connectivity features. They cannot be used with or in place of wires.

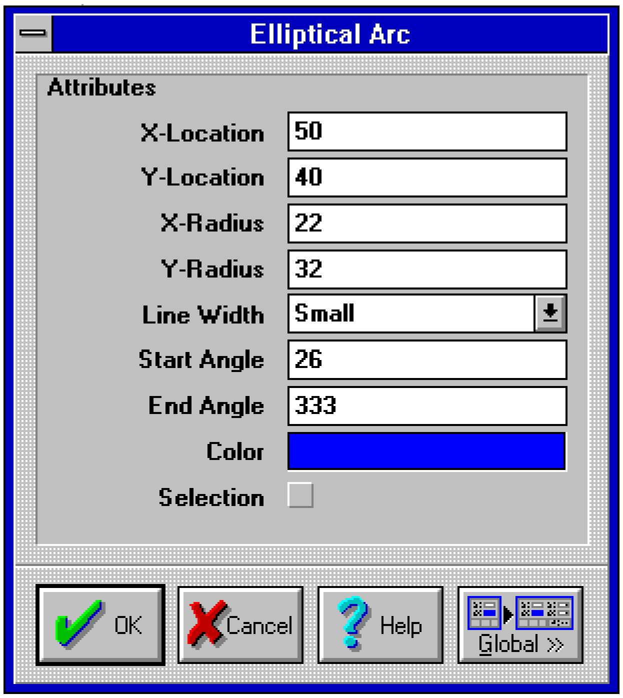
See also

Place Arcs
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 adding reference information to a worksheet, such as building graphical symbols, custom sheet borders, or title blocks.





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 Advanced Schematic 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 Advanced Schematic 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 Advanced Schematic 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 Advanced Schematic 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. Advanced Schematic 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.

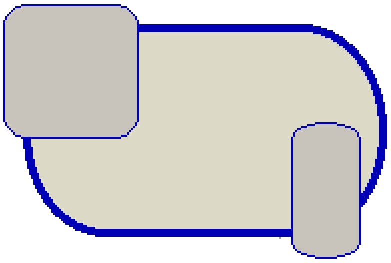
Selections are generally made three ways; using the option in a dialog box, using SHIFT+LEFT MOUSE to add (or remove) individual objects to the current selection and by using the Edit Select and De-select commands to define a selection group.

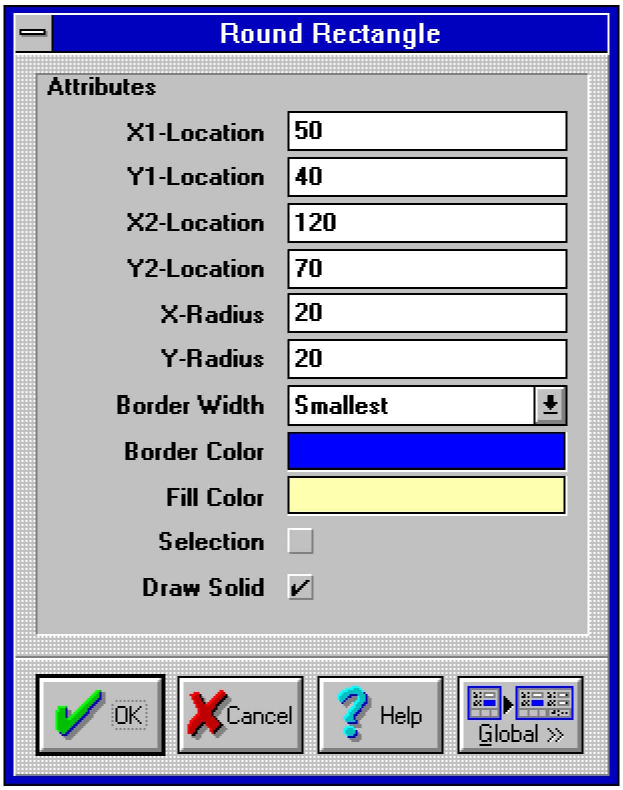
See also

Place Elliptical Arcs

Round Rectangle

Overview Rounded Rectangles are graphical objects with a user-defined line width, start/end coordinates, and fill color.





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.

Advanced Schematic 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.

Advanced Schematic 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. Advanced Schematic 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. Advanced Schematic 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.

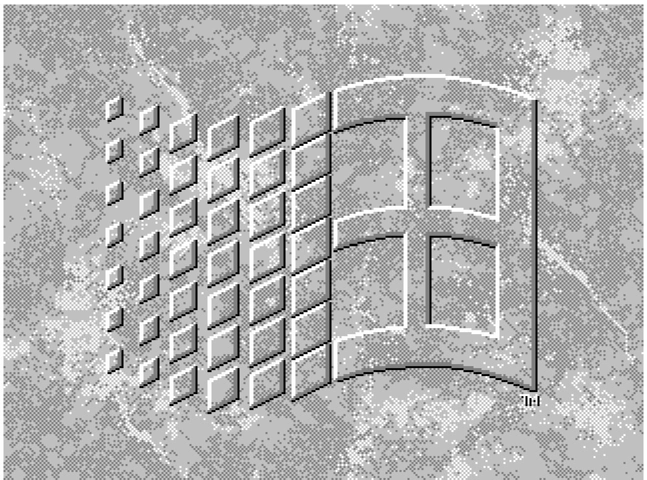
Draw Solid Advanced Schematic allows you to turn the inside area color of Round Rectangles on or off. If this option is on, the inside area of the Rounded Rectangle will be displayed in the fill color. If this option is off, then only the Rounded Rectangles border will be shown.

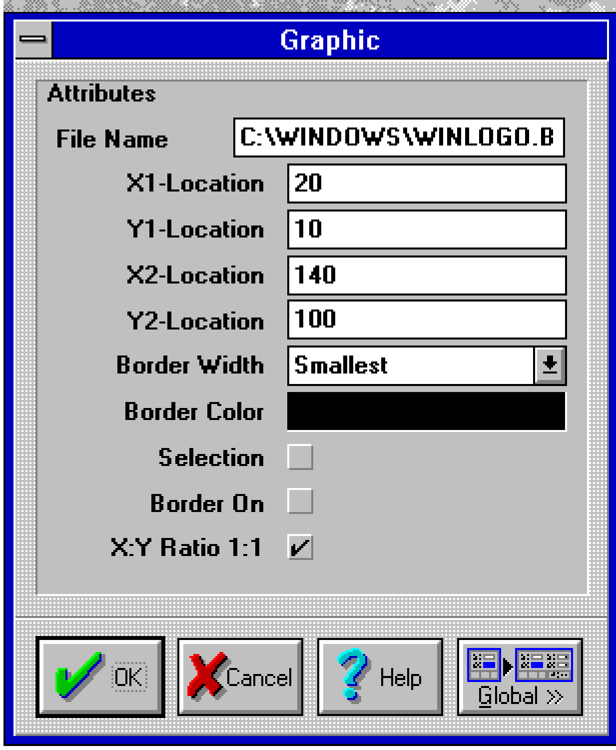
See also

Place Round Rectangle

Graphic

Overview Graphic images are special graphical objects, used for adding reference information to a worksheet, such as company logos, mechanical and electrical symbols or customized 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. Advanced Schematic 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 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.

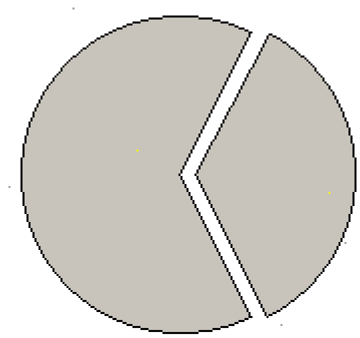
WMF - Only Windows Metafiles which conform to the Aldus Placeable Metafile Format are supported. Most applications which export or import Metafiles support this format.

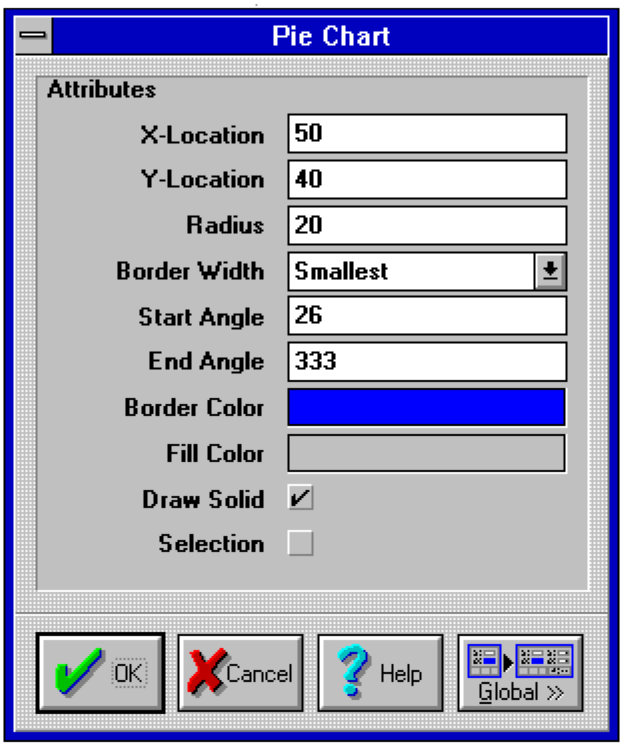
See also

Place Graphic

Pie Chart

Overview Pies are circular graphic objects with a user-defined line width, radius, starting angle and ending angle. Pies are used for adding reference information to a worksheet, such as building graphical symbols, custom sheet borders, or title blocks.





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 Advanced Schematic 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 Advanced Schematic 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 Advanced Schematic 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. Advanced Schematic 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. Advanced Schematic will display all the available colors that your graphics card and monitor supports.

Draw Solid Advanced Schematic 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.

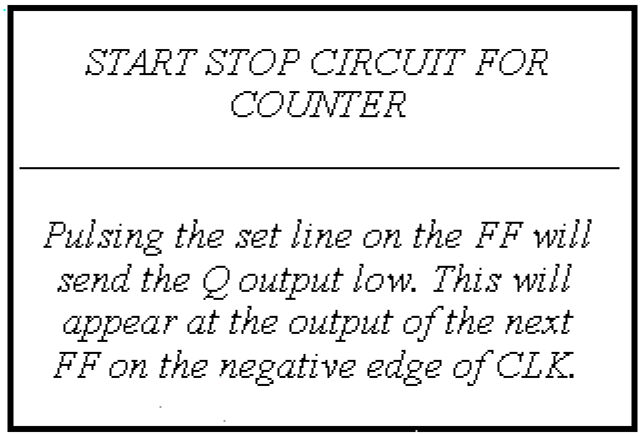
Selections are generally made three ways; using the option in a dialog box, using SHIFT+LEFT MOUSE to add (or remove) individual objects to the current selection and by using the Edit Select and De-select commands to define a selection group.

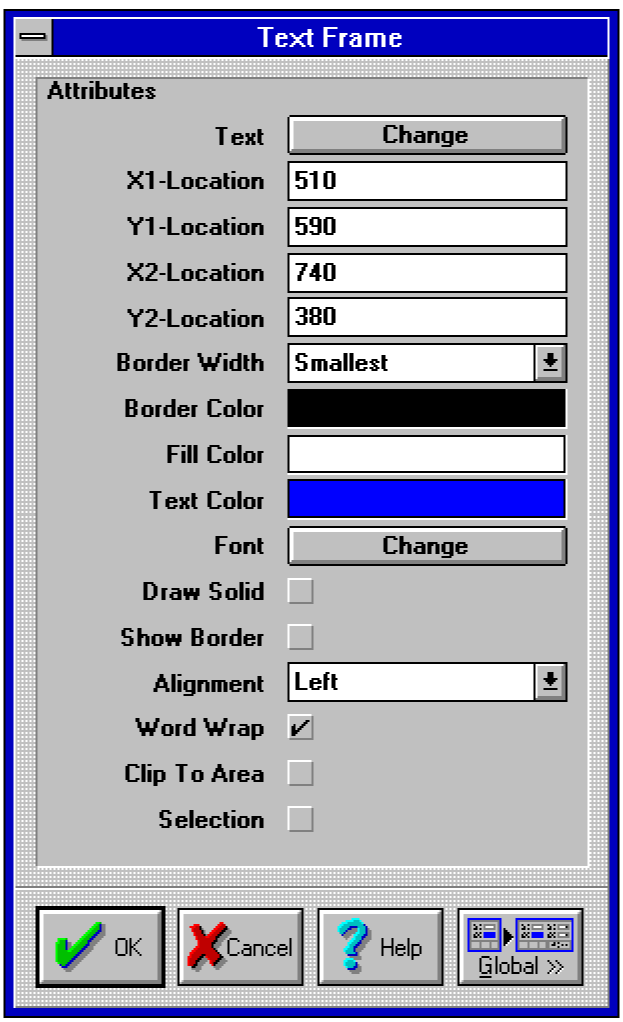
See also

Place Pie Charts

Text Frame

Overview Text frames hold free text. Text frames are used to place detailed notes or descriptive text on the worksheet. Uses might include placing revision history, circuit and logic documentation or some other description on the sheet.





Fields

Text Text frames support up to 32000 characters of text. To add or change text to a text frame click the change button within the text frame dialog box.

X1-Location The left hand edge reference point coordinate of the text frame, along the x-axis.

Y1-Location The bottom edge reference point coordinate of the text frame, along the y-axis.

X2-Location The right hand edge reference point coordinate of the text frame, along the x-axis.

Y2-Location The top edge reference point coordinate of the text frame, along the y-axis.

Border Width Specifies the border outline thickness of Text Frames. There are four selectable border widths; smallest, small, medium and large.

Border Color The surrounding border of Text Frames 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. Advanced Schematic will display all the available colors that your graphics card and monitor supports.

Fill Color The inside area of Text Frames 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. Advanced Schematic will display all the available colors that your graphics card and monitor supports.

Text Color The text within text frames 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. Advanced Schematic will display all the available colors that your graphics card and monitor supports.

Font Text Frames support text formatting, such as, True Type fonts, definable character sizes, italic, bold, underline, and strikeout styles.

Draw Solid Advanced Schematic allows you to turn the inside area color of Text Frames on or off. If this option is on, the inside area of the Text Frame will be displayed in the fill color. If this option is off, then only the Text Frames border (if enabled) and text within the border will be shown.

Show Border The surrounding border of Text Frames can be toggled on or off. If this option is enabled, the border around the Text Frame will be displayed in the border color. If this option is off, then only the text will be shown.

Alignment Text can be aligned within the Text Frame boundary. Three alignment options are provided, Center, Left and Right.

Word Wrap Enable this option to wrap text within the Text Frame boundary.

Clip To Area Enable this option to confine the text within the Text Frame boundary. When enabled, only the text that can fit within the defined text frame border will be displayed. When this option is turned off, the text will flow beyond the text frame and all text will be displayed.

Selection Toggle the selection state of Text Frames by turning this option on or off. If this option is on, the Text Frame 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.

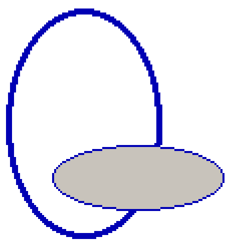
Comments Text Frames support the cut, copy and paste commands for text editing. It is possible to copy text to the Windows clipboard from another application and then paste the text into the text editing window.

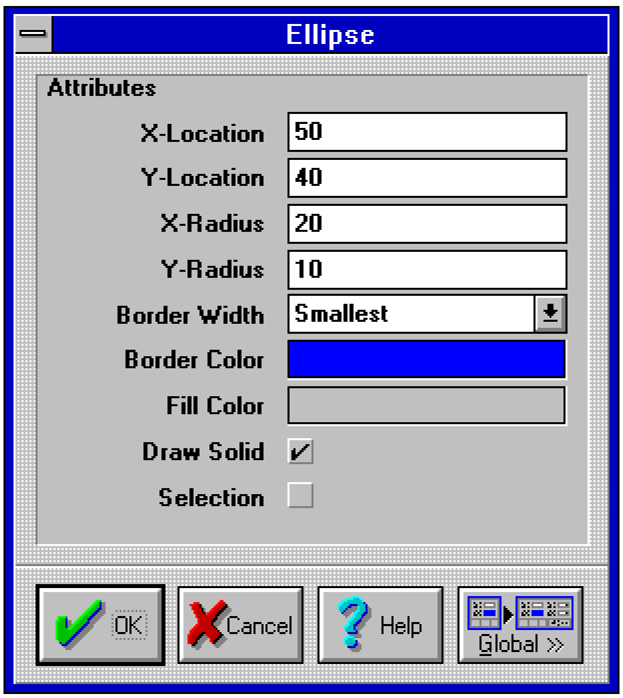
See also

Place Text Frame

Ellipse

Overview Ellipses are graphical objects with a user-defined line width, x and y radius, and fill color. Ellipses are used for adding reference information to a worksheet, such as building graphical symbols, custom sheet borders, or title blocks.





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 Advanced Schematic 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 Advanced Schematic allows you to change the vertical height of Sheet Symbols. 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. Advanced Schematic 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. Advanced Schematic will display all the available colors that your graphics card and monitor supports.

Draw Solid Advanced Schematic 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.

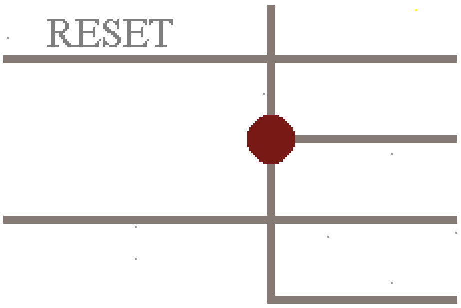
Selections are generally made three ways; using the option in a dialog box, using SHIFT+LEFT MOUSE to add (or remove) individual objects to the current selection and by using the Edit Select and De-select commands to define a selection group.

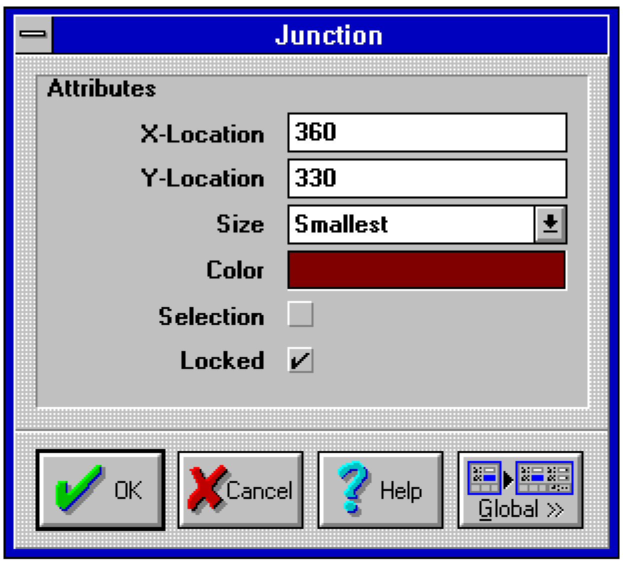
See also

Place Ellipses
Elliptical Arc

Junction

Overview Junctions are small circular objects used to logically join intersecting wires on the schematic.





Fields

X-Location The reference point coordinate of the junction, along the x-axis.

Y-Location The reference point coordinate of the junction, along the y-axis.

Size Specifies the size of junctions. There are four selectable junction sizes; smallest, small, medium and large.

Color Junctions 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. Advanced Schematic will display all the available colors that your graphics card and monitor supports.

Selection Toggle the selection state of Junctions 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.

Locked If checked, the auto-junction feature of Advanced Schematic will not delete the junction.

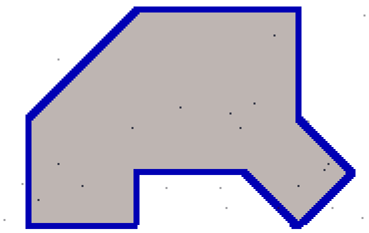
Comments On a worksheet, many wires and buses can either connect or cross. Junctions are placed on the worksheet to distinguish a connection from a cross-over. If you place a junction at any intersection of wires, they will be processed as a single net.

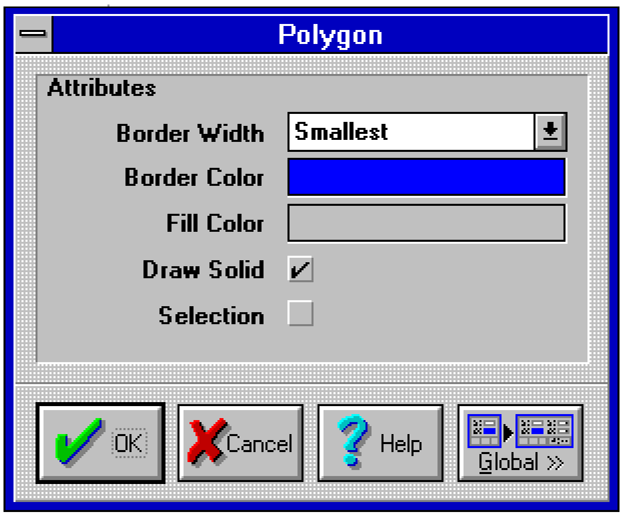
See also

Place Junctions

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 adding reference information to a worksheet, such as building graphical symbols, custom sheet borders, or title blocks.





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. Advanced Schematic 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. Advanced Schematic will display all the available colors that your graphics card and monitor supports.

Draw Solid Advanced Schematic 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.

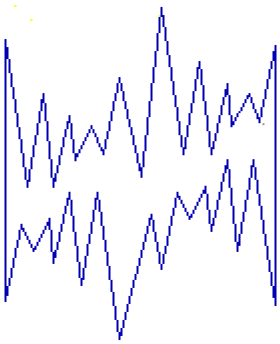
Selections are generally made three ways; using the option in a dialog box, using SHIFT+LEFT MOUSE to add (or remove) individual objects to the current selection and by using the Edit Select and De-select commands to define a selection group.

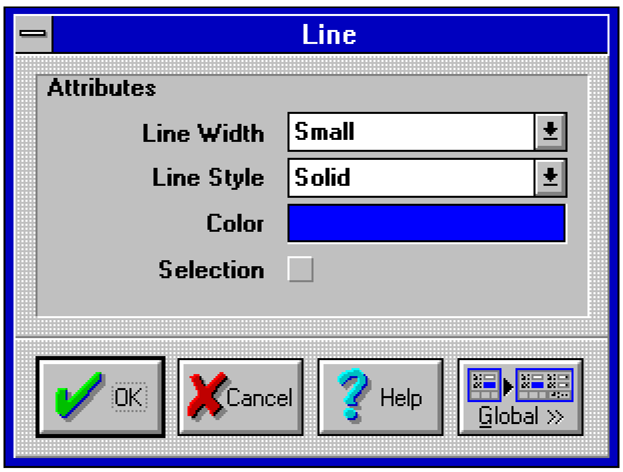
See also

Place Polygons

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 polyline object can be independently moved. Lines are used for adding reference information to a worksheet, such as building graphical symbols, custom sheet borders, or title blocks.





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. Advanced Schematic 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.

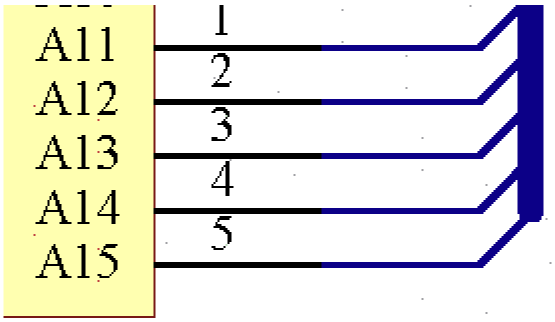
Comments Lines are graphic objects only and do not support Advanced Schematics connectivity features.

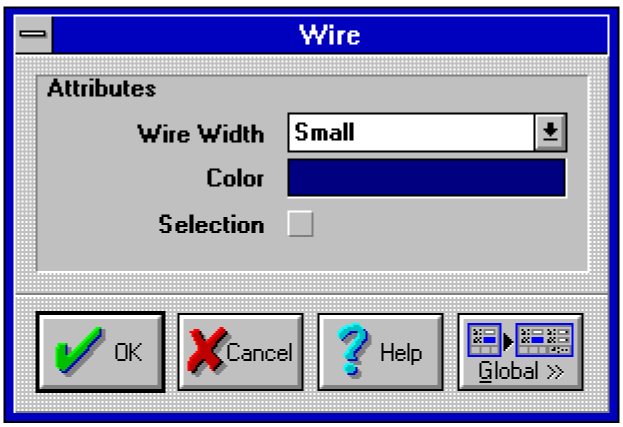
See also

Place Lines

Wire

Overview Wires are straight line segments, which are placed on the worksheet to represent electrical connections.





Fields

Wire Width Specifies the line thickness of the wire segments. There are four selectable wire widths; smallest, small, medium and large.

Color Wire lines can be assigned a color. To assign a new color to the Wire line segments, click in the color box to open the Color Selector dialog box. Advanced Schematic will display all the available colors that your graphics card and monitor supports.

Selection Toggle the selection state of Wires by turning this option on or off. If this option is on, the Wire 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.

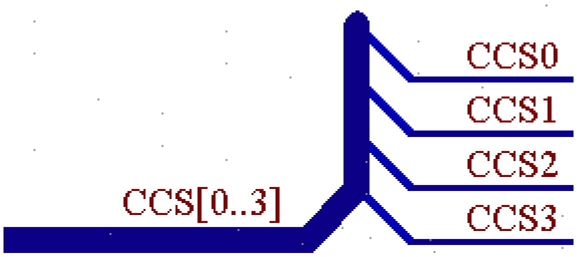
Comments Wires are graphically similar to lines and polylines, but have logical and physical connectivity features.

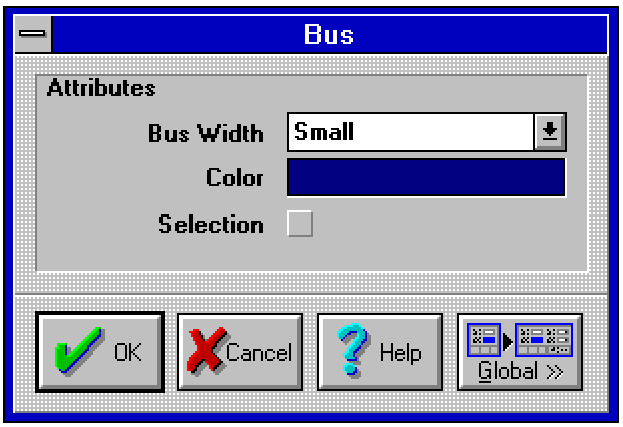
See also

[Place Bus](#)
[Place Bus Entry](#)

Bus

Overview Buses are special graphical objects that represent a common pathway for multiple signals on a worksheet.





Fields

Bus Width Specifies the line thickness of bus segments. There are four selectable bus widths; smallest, small, medium and large.

Color Bus lines can be assigned a color. To assign a new color to the Bus line segment, click in the color box to open the Color Selector dialog box. Advanced Schematic will display all the available colors that your graphics card and monitor supports.

Selection Toggle the selection state of Buses by turning this option on or off. If this option is on, the Buses 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.

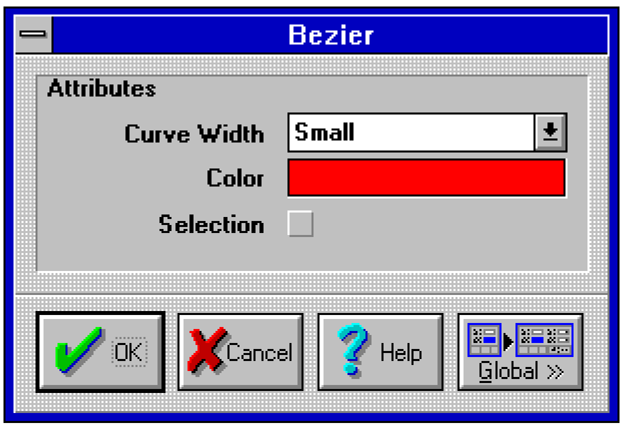
Comments A special graphical primitive, Bus Entry, is needed when connecting a wire to a bus.

See also

[Place Wire](#)
[Bus Entry](#)

Bezier

Overview Beziers are curved line segments, used for adding reference information to a worksheet, such as building graphical, mechanical or electrical 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. Advanced Schematic 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.

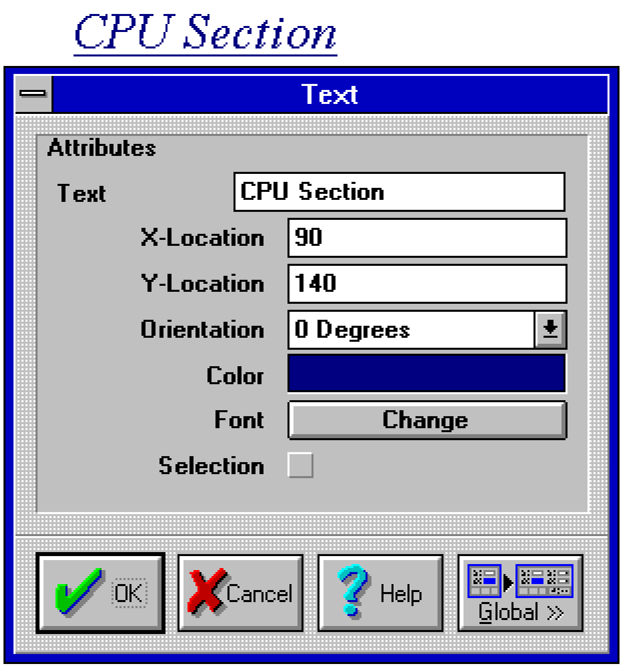
Selections are generally made three ways; using the option in a dialog box, using SHIFT+LEFT MOUSE to add (or remove) individual objects to the current selection and by using the Edit Select and De-select commands to define a selection group.

See also

Place Beziers

Annotation

Overview An annotation is a single line of text used to place notes or comments on the worksheet. Uses might include placing section headings, revision history, timing information or some other description on the sheet.



Fields

Text Text up to 255 characters of text.

X-Location The reference point coordinate of the Annotation, along the x-axis.

Y-Location The reference point coordinate of the Annotation, along the y-axis.

Orientation Annotations 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 an annotation, you can rotate it around the cursor by pressing the SPACEBAR.

Color Annotations can be assigned a color. To assign a new color to the Annotation, click in the color box to open the Color Selector dialog box. Advanced Schematic will display all the available colors that your graphics card and monitor supports.

Font Annotations support text formatting, such as, True Type fonts, definable character sizes, italic, bold, underline, and strikeout styles.

Selection Toggle the selection state of Annotations by turning this option on or off. If this option is on, the Annotation will be outlined in the selection color defined in the Options Preferences dialog box.

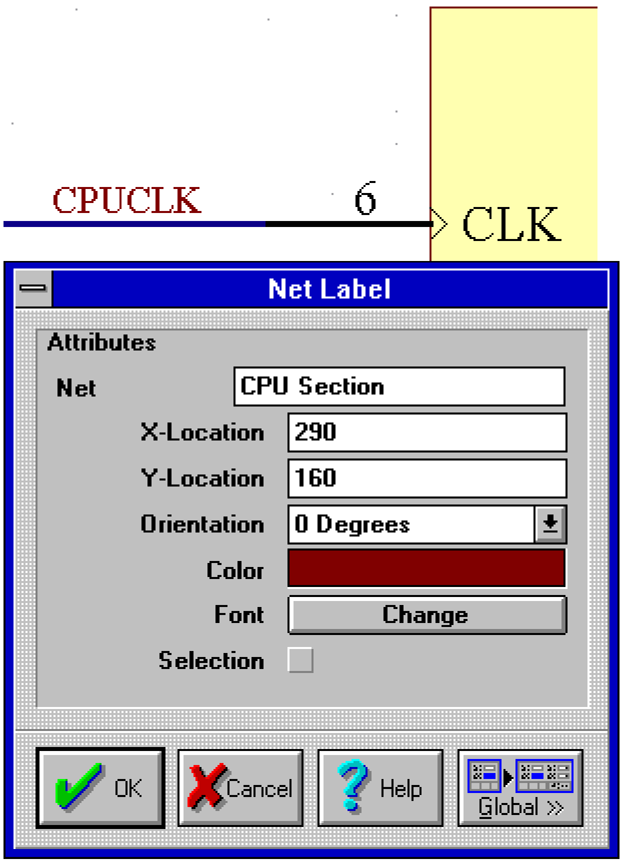
Selections are generally made three ways; using the option in a dialog box, using SHIFT+LEFT MOUSE to add (or remove) individual objects to the current selection and by using the Edit Select and De-select commands to define a selection group.

See also

Place Annotation
Text Frame

Net Label

Overview Net Labels assign a wire and its entire physical net to a specific net name. Net Labels are also used to connect signals (wires) together without actually physically connecting them. You can place Net Labels horizontally or vertically on a worksheet.



Fields

Net Net Labels support up to 255 characters of text. The label will be used as the identifier for connecting to other signals in the worksheet or across multiple sheets in a project.

X-Location The reference point coordinate of the Net Label, along the x-axis.

The reference point coordinate of the Annotation, along the x-axis.

Y-Location The reference point coordinate of the Net Label, along the y-axis.

The reference point coordinate of the Annotation, along the y-axis.

Orientation Net labels 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 a net label, you can rotate it around the cursor by pressing the SPACEBAR.

Annotations 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 an annotation, you can rotate it around the cursor by pressing the SPACEBAR.

Color Net Labels can be assigned a color. To assign a new color to the net name, click in the color box to open the Color Selector dialog box. Advanced Schematic will display all the available colors that your graphics card and monitor supports.

Annotations can be assigned a color. To assign a new color to the Annotation, click in the color box to open the Color Selector dialog box. Advanced Schematic will display all the available colors that your graphics card and monitor supports.

Font Net labels support text formatting, such as, True Type fonts, definable character sizes, italic, bold, underline, and strikeout styles.

Annotations support text formatting, such as, True Type fonts, definable character sizes, italic, bold, underline, and strikeout styles.

Selection Toggle the selection state of Net Labels by turning this option on or off. If this option is on, the Net Label will be outlined in the selection color defined in the Options Preferences dialog box.

Text up to 255 characters of text.

Toggle the selection state of Annotations by turning this option on or off. If this option is on, the Annotation 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.

Comments The Net Label must be placed directly on the wire or bus. The snap grid can be used to ensure that the label is in a legal position. All Net Labels with the same name are considered to be electrically connected and signals connected externally to other worksheets in a project may use Net Labels and/or Ports.

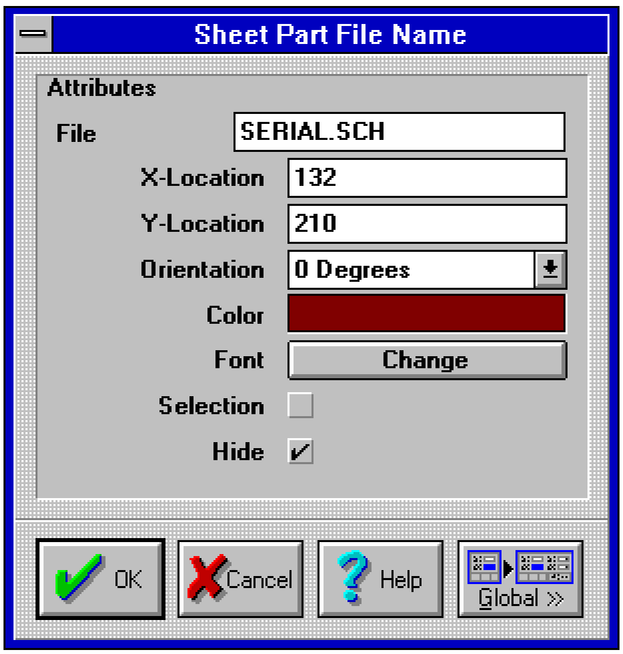
An inversion bar may be placed over net label characters by typing a backslash \ after each character. For example, to place an inversion bar over the net label 'ENABLE', type 'E\N\A\B\L\\'. To place the inversion bar over the first two characters, type 'E\N\ABLE'.

See also

Place Net Label
Port

Sheet Part File Name

Overview Sheet parts represent another schematic worksheet, the Sheet Part Filename is the link to the schematic worksheet.



Fields

X-Location The reference point coordinate of the sheet part filename, along the x-axis.

Y-Location The reference point coordinate of the sheet part filename, along the y-axis.

Orientation Sheet part filenames can be rotated 360 degrees using 90 degree increments. There are four selectable orientations, 0 degrees, 90 degrees, 180 degrees and 270 degrees. While moving one of the fields, you can rotate it around the cursor by pressing the SPACEBAR.

Color Sheet part filenames can be assigned a color. To assign a new color to a sheet part filename, click in the color box to open the Color Selector dialog box. Advanced Schematic will display all the available colors that your graphics card and monitor supports.

Font Sheet part filenames support text formatting, such as, True Type fonts, definable character sizes, italic, bold, underline, and strikeout styles.

Selection Toggle the selection state of the Sheet part filename by turning this option on or off. If this option is on, the field 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.

Hide If this option is enabled, the Sheet part filename is hidden and is not displayed on the worksheet. By default this option is enabled.

Comments Similar to Sheet Symbols, however can be defined with any graphic representation in the library editor, the parts pins act as sheet entries. You can use the hierarchy, Up, Down, Create Sheet from Symbol commands with Sheet Parts.

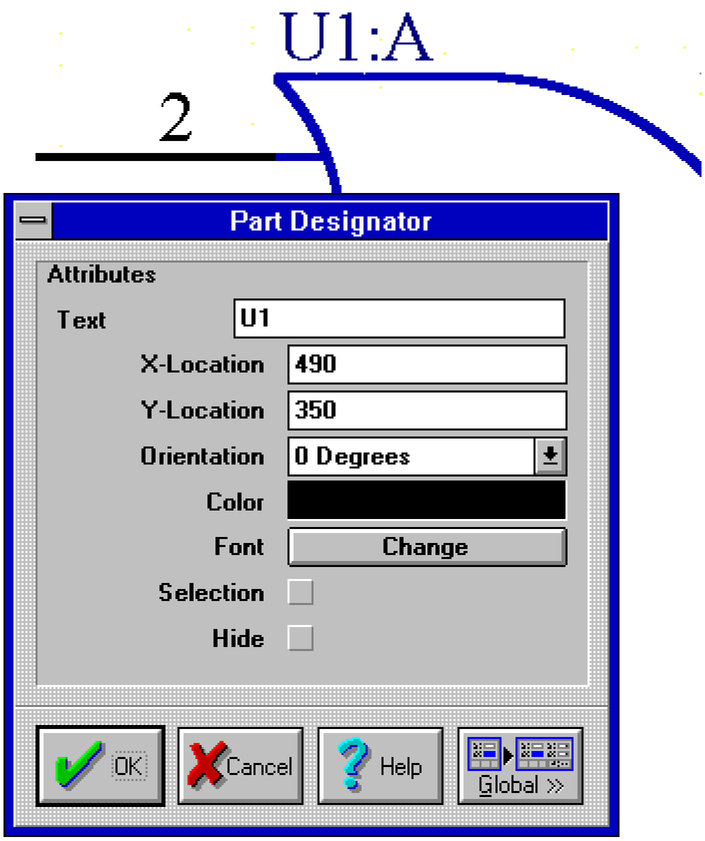
Note. The default sheet Part Filename is an asterix (*), this can cause file loading problems when exporting the schematic file back to Orcad format. Make sure to remove all asterix symbols from the sheet Part filename fields before saving and loading the file into Orcad. You can globally edit these fields.

See also

Part
Sheet Symbol
Sheet Symbol File Name

Part Designator

Overview The part designator is the name identifier for each part placed on the schematic worksheet. The designator may be changed by either double clicking on this field or using the Rename button in the Component Browser.



Fields

Text Part designators may have up to 255 characters.

X-Location The reference point coordinate of the part designator, along the x-axis.

Y-Location The reference point coordinate of the part designator, along the y-axis.

Orientation Part designators can be rotated 360 degrees using 90 degree increments. There are four selectable orientations, 0 degrees, 90 degrees, 180 degrees and 270 degrees. While moving the part designator, you can rotate it around the cursor by pressing the SPACEBAR.

Color Part designators can be assigned a color. To assign a new color to a part designator, click in the color box to open the Color Selector dialog box. Advanced Schematic will display all the available colors that your graphics card and monitor supports.

Font Part designators support text formatting, such as, True Type fonts, definable character sizes, italic, bold, underline, and strikeout styles.

Selection Toggle the selection state of the part designator by turning this option on or off. If this option is on, the part designator 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.

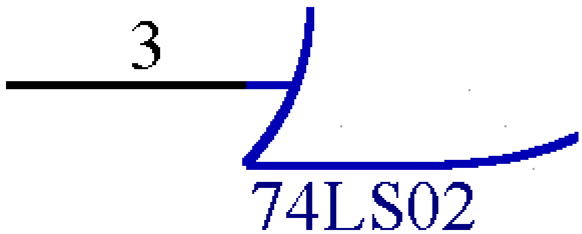
Hide If this option is enabled, the designator is hidden and is not displayed on the worksheet.

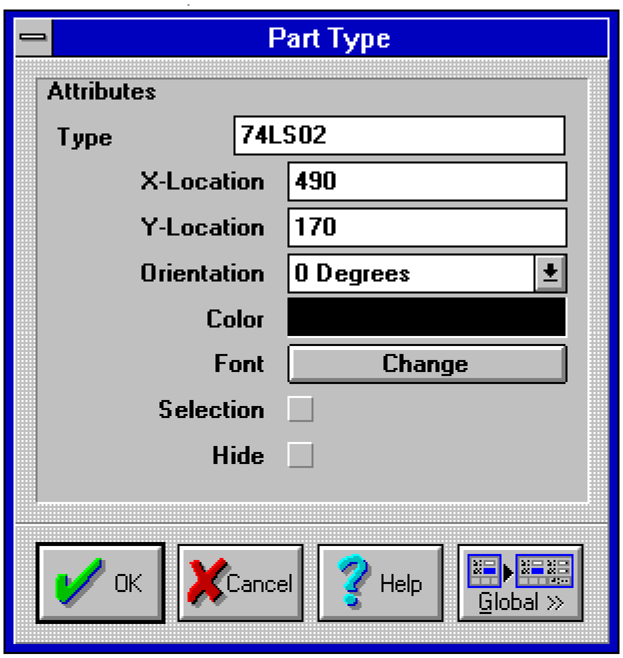
See also

Part

Part Type

Overview The Part Type is the name of the component on the schematic worksheet. For example, 10k can be chosen as the part type for the resistor.





Fields

Type Part names may have up to 255 characters.

X-Location The reference point coordinate of the part type, along the x-axis.

Y-Location The reference point coordinate of the part type, along the y-axis.

Orientation Part type names can be rotated 360 degrees using 90 degree increments. There are four selectable orientations, 0 degrees, 90 degrees, 180 degrees and 270 degrees. While moving the part type name, you can rotate it around the cursor by pressing the SPACEBAR.

Color Part type names can be assigned a color. To assign a new color to a part type, click in the color box to open the Color Selector dialog box. Advanced Schematic will display all the available colors that your graphics card and monitor supports.

Font Part type names support text formatting, such as, True Type fonts, definable character sizes, italic, bold, underline, and strikeout styles.

Selection Toggle the selection state of the part type by turning this option on or off. If this option is on, the part type 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.

Hide If this option is enabled, the part type is hidden and is not displayed on the worksheet.

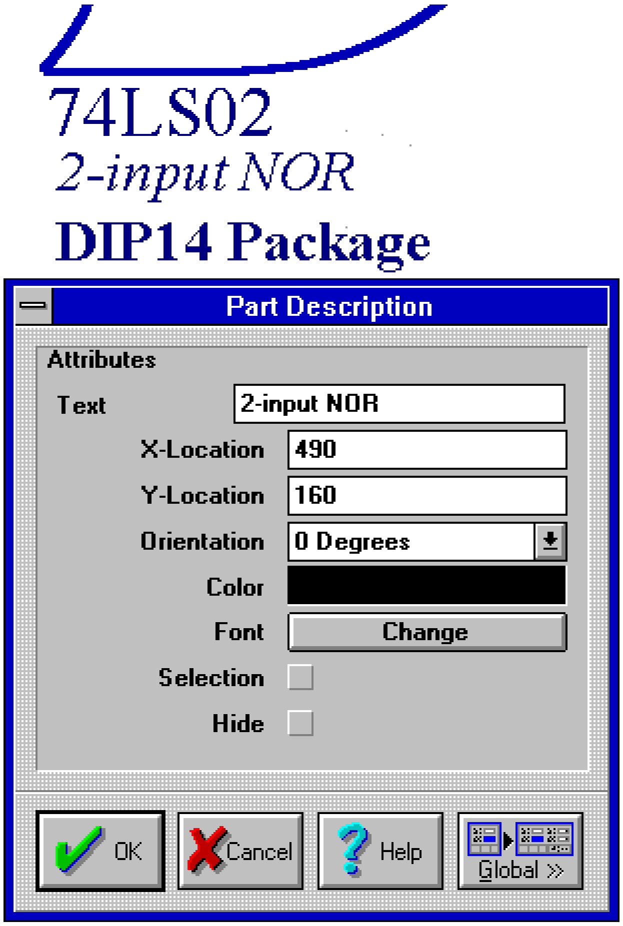
Comments You can move and position the part type anywhere on the worksheet, if you move the part later, the type name will move with the part.

See also

Part

Part Description

Overview Parts have eight extra description fields that are used for adding reference information to a part on the schematic worksheet or for inclusion in a Bill of Materials file. The fields can be moved and positioned individually. They also remain associated with the part, so if you move the part later, the fields will move with the part.



Fields

Text Each description field can be 255 characters long.

X-Location The reference point coordinate of the description field, along the x-axis.

Y-Location The reference point coordinate of the description field, along the y-axis.

Orientation Part description fields can be rotated 360 degrees using 90 degree increments. There are four selectable orientations, 0 degrees, 90 degrees, 180 degrees and 270 degrees. While moving one of the fields, you can rotate it around the cursor by pressing the SPACEBAR.

Color Part description names can be assigned a color. To assign a new color to a description field, click in the color box to open the Color Selector dialog box. Advanced Schematic will display all the available colors that your graphics card and monitor supports.

Font Part description fields support text formatting, such as, True Type fonts, definable character sizes, italic, bold, underline, and strikeout styles.

Selection Toggle the selection state of the description field by turning this option on or off. If this option is on, the field 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.

Hide If this option is enabled, the description field is hidden and is not displayed on the worksheet. By default this option is enabled.

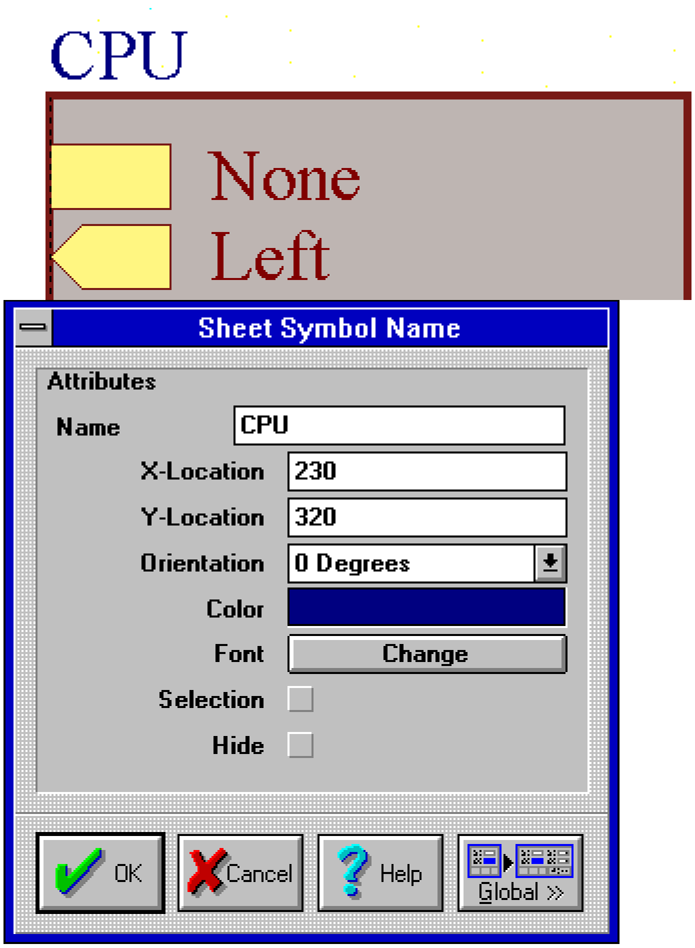
Comments These eight description fields belong to this instance of this part only. They are in additional to the eight text fields that are defined for a specific component in the library editor.

See also

Part

Sheet Symbol Name

Overview The Sheet symbol name is a description field used to label a sheet symbol.



Fields

Name Sheet symbol names support up to 255 characters of text.

X-Location The reference point coordinate of the sheet symbol file name, along the x-axis.

Y-Location The reference point coordinate of the sheet symbol file name, along the y-axis.

Orientation Sheet symbol names can be rotated 360 degrees using 90 degree increments. There are four selectable orientations, 0 degrees, 90 degrees, 180 degrees and 270 degrees. While moving the name, you can rotate it around the cursor by pressing the SPACEBAR.

Color Sheet symbol names can be assigned a color. To assign a new color to the file name, click in the color box to open the Color Selector dialog box. Advanced Schematic will display all the available and dithered colors that your graphics card and monitor supports.

Font Sheet symbol names support text formatting, such as, True Type fonts, definable character sizes, italic, bold, underline, and strikeout styles.

Selection Toggle the selection state of the sheet symbol name by turning this option on or off. If this option is on, the name 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.

Hide If this option is enabled, the sheet symbol name is hidden and is not displayed on the worksheet.

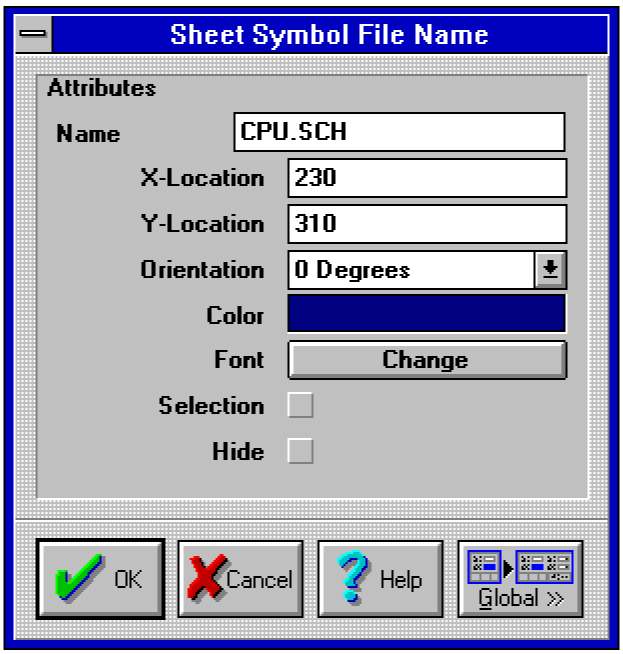
Comments You can move and position the sheet symbol name anywhere on the worksheet, if you move the sheet symbol later, the name will move with the sheet symbol.

See also

Sheet Symbol

Sheet Symbol File Name

Overview Sheet Symbols represent another schematic worksheet, the file name is the link to the schematic worksheet.



Fields

Name The name of the schematic worksheet file, including the full path. (e.g., C:\SCHFILES\SHEET1.SCH)

X-Location The reference point coordinate of the sheet symbol file name, along the x-axis.

Y-Location The reference point coordinate of the sheet symbol file name, along the y-axis.

Orientation Sheet symbol file names can be rotated 360 degrees using 90 degree increments. There are four selectable orientations, 0 degrees, 90 degrees, 180 degrees and 270 degrees. While moving the file name, you can rotate it around the cursor by pressing the SPACEBAR.

Color Sheet Symbol file names can be assigned a color. To assign a new color to the file name, click in the color box to open the Color Selector dialog box. Advanced Schematic will display all the available colors that your graphics card and monitor supports.

Font Sheet symbol file names support text formatting, such as, True Type fonts, definable character sizes, italic, bold, underline, and strikeout styles.

Selection Toggle the selection state of the sheet symbol file name by turning this option on or off. If this option is on, the file name 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.

Hide If this option is enabled, the sheet symbol file name is hidden and is not displayed on the worksheet.

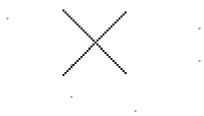
Comments You can move and position the file name anywhere on the schematic worksheet, if you move the sheet symbol later, the file name will move with the sheet symbol.

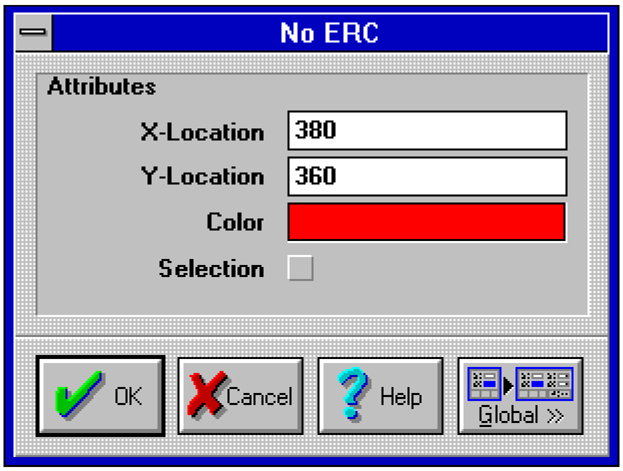
See also

Sheet Symbol
Sheet Part File Name

No ERC

Overview No ERC are special symbols that identify a pin on a part that is to be left unconnected. During electrical design rules checking, unconnected pins with No ERC symbols will be ignored.





Fields

X-Location The reference point coordinate the No ERC symbol, along the x-axis.

Y-Location The reference point coordinate of a No ERC symbol, along the y-axis.

Color No ERC symbols can be assigned a color. To assign a new color to the No ERC symbol, click in the color box to open the Color Selector dialog box. Advanced Schematic will display all the available colors that your graphics card and monitor supports.

Selection Toggle the selection state of No ERC symbols by turning this option on or off. If this option is on, the No ERC symbol will be outlined in the selection color defined in the Options Preferences dialog box.

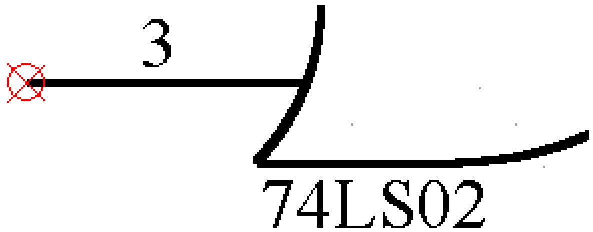
Selections are generally made three ways; using the option in a dialog box, using SHIFT+LEFT MOUSE to add (or remove) individual objects to the current selection and by using the Edit Select and De-select commands to define a selection group.

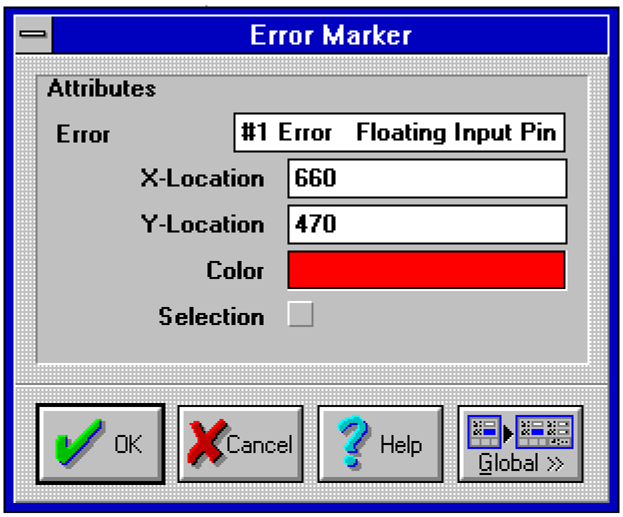
See also

Place No ERC
Jump To Next Error

Error Marker

Overview Error Markers are special symbols that flag ERC error violations, generated after an Electrical Rules Check report.





Fields

Error The description of the error violation. This error and description is also written to the ERC report file.

X-Location The reference point coordinate of the Error Marker, along the x-axis.

Y-Location The reference point coordinate of the Error Marker, along the y-axis.

Color The color of the Error Marker flag, defined in the Options Preferences dialog box.

Selection Toggle the selection state of Error Markers by turning this option on or off. If this option is on, the Error Marker will be outlined in the selection color defined in the Options Preferences dialog box.

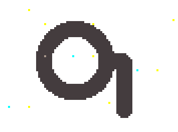
Selections are generally made three ways; using the option in a dialog box, using SHIFT+LEFT MOUSE to add (or remove) individual objects to the current selection and by using the Edit Select and De-select commands to define a selection group.

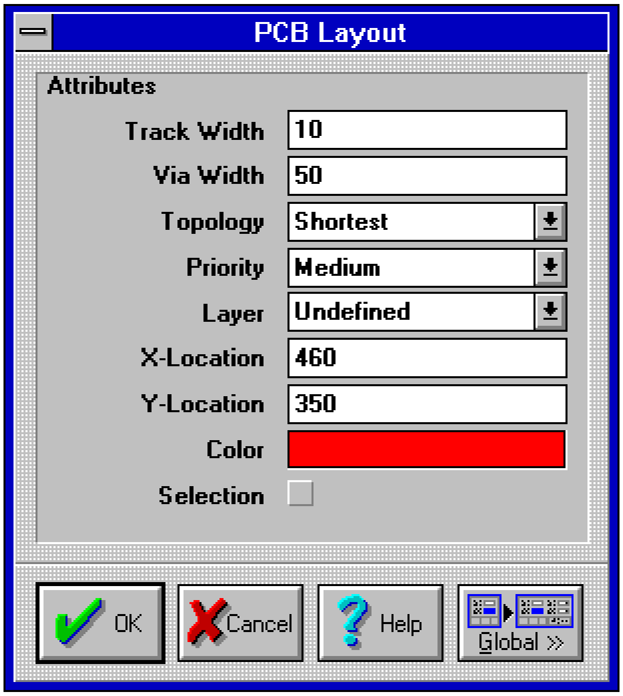
See also

Electrical Rules Check

PCB Layout

Overview PCB Layouts are special symbols that allow you to attach board layout information to a specific net. This information is saved into the netlist file which is later passed to Advanced PCB, OrCAD PCB or other compatible board layout systems.





Fields

Track Width Sets the routing track width that will be used by a board layout program to auto route the net flagged by this PCB layout directive. The width can be any integer (whole number) value, the default is 10mil. Protel for Windows Advanced PCB allows track widths of 1 to 9999mil. Refer to your board layout documentation for details on maximum and minimum track routing widths.

Via Width Sets the routing via diameter that will be used by a board layout program to auto route the net flagged by this PCB layout directive. The width can be any integer (whole number) value, the default is 50mil. Protel for Windows Advanced PCB allows track widths of 1 to 9999mil. Refer to your board layout documentation for details on maximum and minimum track routing widths.

Topology Sets the optimization method that will be used by a board layout program to sort the connections flagged by this PCB layout directive. Six selectable optimization methods are available (all supported by Advanced PCB); X-Bias, Y-Bias, Shortest, Daisy Chain, Minimum Daisy Chain and Start/End Daisy Chain, the default is Shortest. Refer to your board layout documentation for details on supported optimization methods.

Priority Sets the routing priority that will be used by a board layout program to auto route the net flagged by this PCB layout directive. Five selectable routing priorities are available (all supported by Advanced PCB); Highest, High, Medium, Low and Lowest, the default is Medium. Refer to your board layout documentation for details on supported routing priorities.

Layer Sets the routing track layer that will be used by a board layout program to auto route the net flagged by this PCB layout directive. Thirty four selectable routing track layers are available (all supported by Protel for Windows Advanced PCB). Undefined will not assign a routing track layer to the net, the board layout program will use any available layers when auto routing this net. Refer to your board layout documentation for details on supported track routing layers.

X-Location The reference point coordinate of the PCB layout directive, along the x-axis.

Y-Location The reference point coordinate of the PCB layout directive, along the y-axis.

Color PCB Layout directives can be assigned a color. To assign a new color to the PCB Layout directive, click in the color box to open the Color Selector dialog box. Advanced Schematic will display all the available colors that your graphics card and monitor supports.

Selection Toggle the selection state of Board Layout Directives by turning this option on or off. If this option is on, the Board Layout Directive will be outlined in the selection color defined in the Options Preferences dialog box.

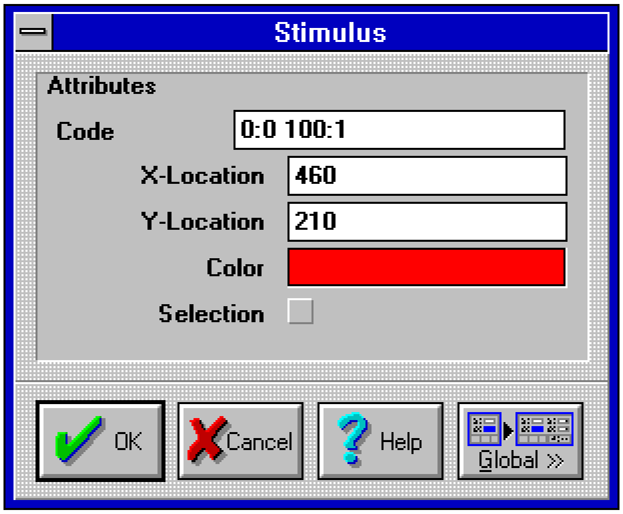
Selections are generally made three ways; using the option in a dialog box, using SHIFT+LEFT MOUSE to add (or remove) individual objects to the current selection and by using the Edit Select and De-select commands to define a selection group.

See also

Place PCB Layout

Stimulus

Overview A Stimulus is a special symbol which is used to identify a node or net to be stimulated when the digital simulation is run.



Fields

Code The stimulus value is an expression describing the pattern of states present in the circuit logic.

X-Location The reference point coordinate of the Stimulus, along the x-axis.

Y-Location The reference point coordinate of the Stimulus, along the y-axis.

Color A Stimulus can be assigned with a color. To assign a new color to the Stimulus, click in the color box to open the Color Selector dialog box. Advanced Schematic will display all the available colors that your graphics card and monitor supports.

Selection Toggle the selection state of Stimulus objects by turning this option on or off. If this option is on, the Stimulus 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.

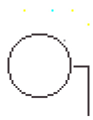
Comments Stimulus symbols are used in OrCAD digital simulation tools, Refer to your OrCAD documentation for further information about using simulation stimulus.

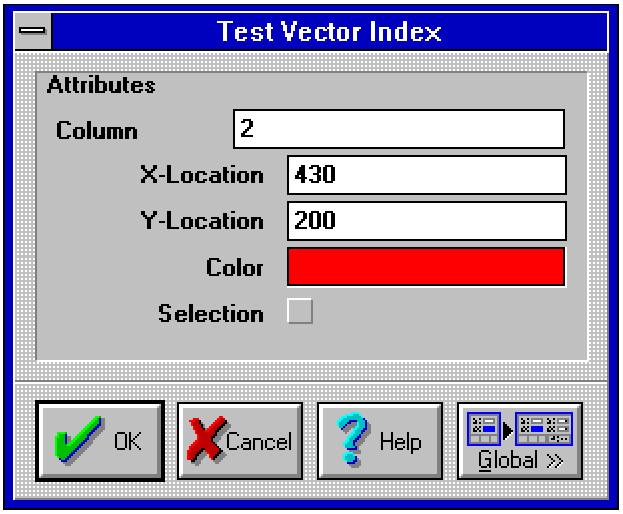
See also

Test Vector Index
Probe
Place Stimulus

Test Vector Index

Overview Test Vectors are special symbols used to identify a node with a simulation test vector. The test vectors are referred to by a column number, which indicates the column of the test vector file to use when the simulation is run.





Fields

Column Name of the Test Vector Index, this indicates which column of the test vector file to use when the simulation is run.

X-Location The reference point coordinate of the Test Vector Index, along the x-axis.

Y-Location The reference point coordinate of the Test Vector Index, along the y-axis.

Color Test Vectors can be assigned a color. To assign a new color to the Test Vector, click in the color box to open the Color Selector dialog box. Advanced Schematic will display all the available colors that your graphics card and monitor supports.

Selection Toggle the selection state of Test Vector Indexes by turning this option on or off. If this option is on, the Test Vector Index 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.

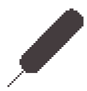
Comments Test Vector Index symbols are used in OrCAD digital simulation tools, refer to your OrCAD documentation for further information about using simulation vectors.

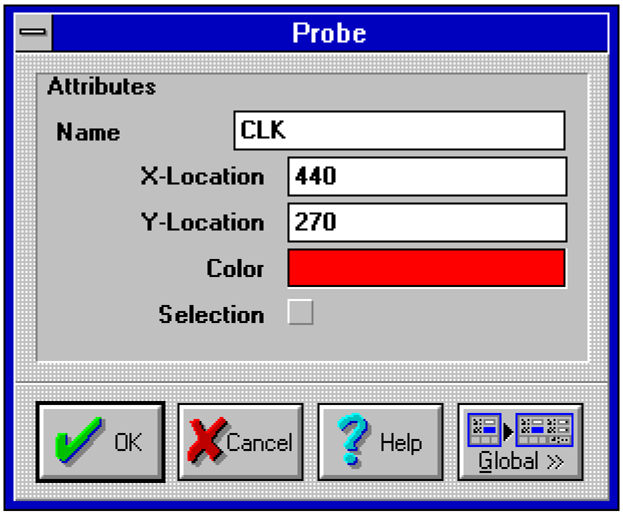
See also

Place Test Vector Index
Probe
Stimulus

Probe

Overview A Probe is a special marker which is placed on the worksheet to identify nodes for digital simulation.





Fields

Name The name that is used when tracing the timing of the circuit in digital simulation.

X-Location The reference point coordinate of a Probe, along the x-axis.

Y-Location The reference point coordinate of a Probe, along the y-axis.

Color Probes can be assigned a color. To assign a new color to the Probe, click in the color box to open the Color Selector dialog box. Advanced Schematic will display all the available colors that your graphics card and monitor supports.

Selection Toggle the selection state of Probes by turning this option on or off. If this option is on, the Probe 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.

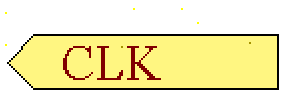
Comments Probes are used in OrCAD digital simulation tools, refer to your OrCAD documentation for further information about using simulation probes.

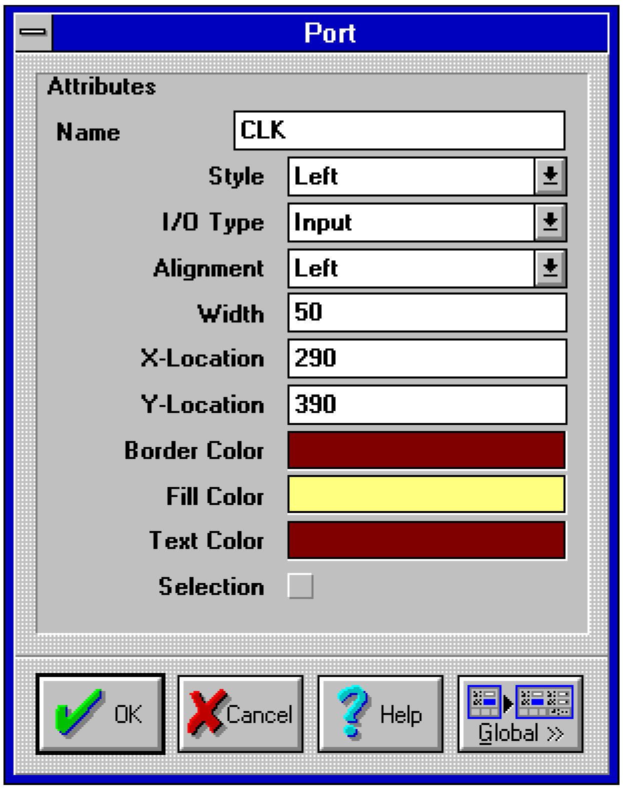
See also

Place Probe
Test Vector Index
Stimulus

Port

Overview Ports are special symbols that are used to direct a signal (wire or bus) to another schematic sheet, they may also be used internally within a sheet. All Ports with the same name are considered to be electrically connected.





Fields

Name Port names or labels support up to 255 characters of text. The label will be used as the identifier for connecting to signals on other sheets in a flat sheet design.

Style Four display styles are available for Ports, left arrow, right arrow, left and right arrow and none. A Ports 'Style' is independent from its I/O Type. For example, a Port with the Style of 'Left' does not necessarily mean the Port is 'Output'.

I/O Type Specifies the type of signal processing characteristics of the port. There are four I/O types, Input, Output, Bi-directional and Unspecified. The Electrical Rules Check recognizes Sheet Entry types and reports violations.

Alignment Port names can be aligned within the Port boundary. Three alignment options are provided, Center, Left and Right.

Width Advanced Schematic allows you to change the horizontal width of Ports. The width can be any integer (whole number) value. A value of 1 = 10mil = .254mm. When the option 'OrCAD Ports' is enabled in the Options Preferences dialog box, the length of all Ports in a schematic project are re-sized to the OrCAD equivalent and the length of the Port is restricted from being manually edited. This option is used when exporting a schematic file out to the OrCAD format. First, enable this option, then manually clean-up any Port-to-Wire connections that may have been disconnected.

X-Location The reference point coordinate of a Port, along the x-axis.

Y-Location The reference point coordinate of a Port, along the y-axis.

Border Color The surrounding border of Ports 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. Advanced Schematic will display all the available colors that your graphics card and monitor supports.

Fill Color The inside area of Ports 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. Advanced Schematic will display all the available colors that your graphics card and monitor supports.

Text Color Port names or labels can be assigned a color. To assign a new color to the Name, click in the color box to open the Color Selector dialog box. Advanced Schematic will display all the available colors that your graphics card and monitor supports.

Selection Toggle the selection state of Ports by turning this option on or off. If this option is on, the Port 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.

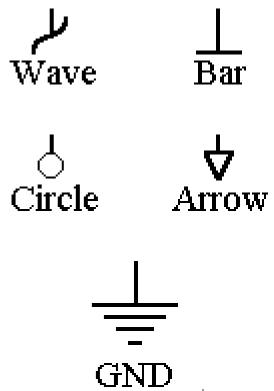
Comments When exporting a schematic project to OrCAD format, make sure to enable the option "OrCAD Ports" in the Options Preferences dialog box. Note: Manual clean-up of Port-to-Wire connections may be necessary, if this option was disabled during the course of design.

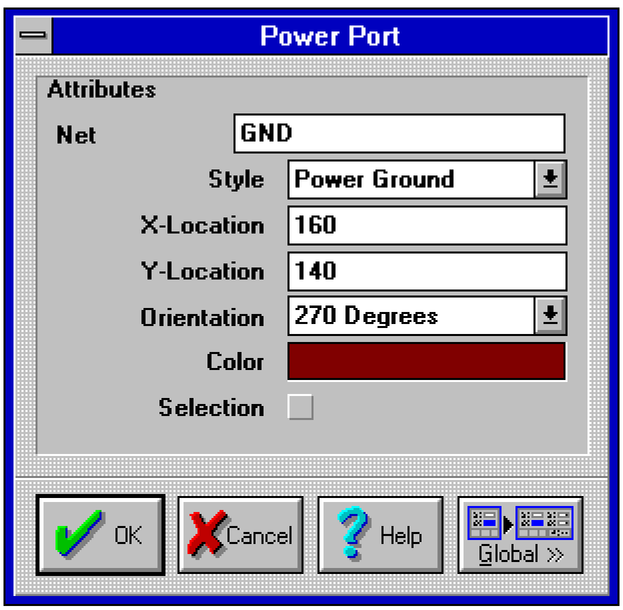
See also

Place Port
Net Label

Power Port

Overview Power Ports are special symbols that represent a power supply net. These allow you to indicate a power net conveniently at various locations on the sheet by connecting Power Ports to pins or to the end of wires.





Fields

Net Power Port names or labels support up to 255 characters of text. The Label will be used as the identifier for connecting to signals on other sheets in a flat or hierarchical design. Common used names include, +5, VCC, -12, GND, VSS etc.

Style Power Ports can be represented by five different styles; Circle, Arrow, Bar, Wave and Ground. The styles are graphical options only and have no electrical properties.

X-Location The reference point coordinate of a Power Port, along the x-axis.

Y-Location The reference point coordinate of a Power Port, along the y-axis.

Orientation Power Ports 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 a Power Port, you can rotate it around the cursor by pressing the SPACEBAR.

Color Power Ports can be assigned a color. To assign a new color to the Power Port, click in the color box to open the Color Selector dialog box. Advanced Schematic will display all the available colors that your graphics card and monitor supports.

Selection Toggle the selection state of Power Ports by turning this option on or off. If this option is on, the Power Port 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.

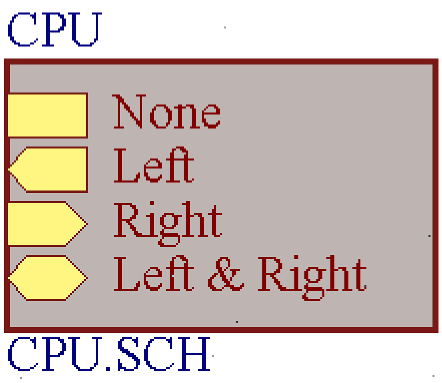
Comments When generating a netlist, signals that are connected directly to Power Ports are automatically tied to the name of the Power Port.

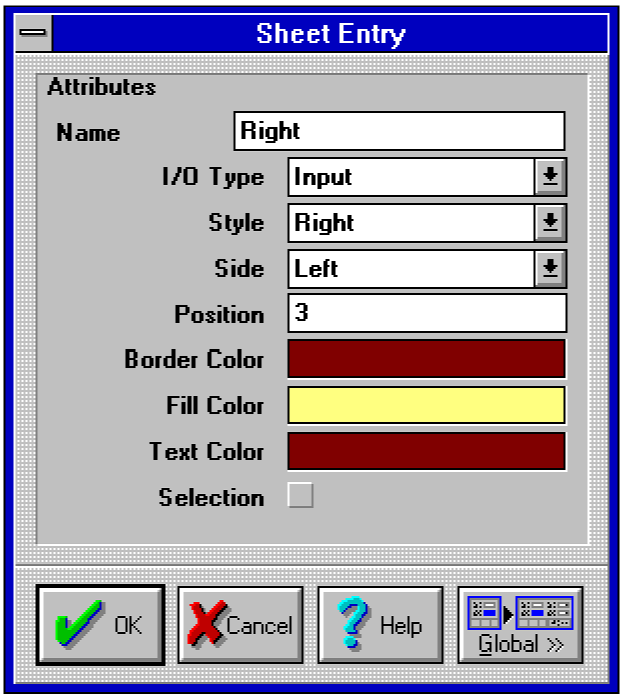
See also

Place Power Port

Sheet Entry

Overview Sheet Entry symbols direct signals to another sheet in a hierarchical design.





Fields

Name Sheet Entry Names or Labels support up to 255 characters of text. The Label will be used as the identifier for connecting to signals on other sheets in the hierarchy.

I/O Type Specifies the type of signal processing characteristics of the sheet entry. There are four I/O types, Input, Output, Bi-directional and Unspecified. The Electrical Rules Check (ERC) recognizes Sheet Entry types and reports violations. A Sheet Entry will connect to a port on the sheet represented by the Sheet Symbol. Although the ERC is programmable and connection errors can have their interpretation altered, the following rules should be followed.

if Sheet Entry is input, the connecting port should be output.

if sheet Entry is output, the connecting port should be input.

if sheet Entry is Bi-directional, the port should be Bi-directional.

if sheet Entry is Unspecified, the port should be Unspecified.

Style Four display styles are available for Sheet Entry symbols, left arrow, right arrow, left and right arrow and none. A Sheet Entry symbols 'Style' is independent from its Electrical type. For example, a Sheet Entry with the Style of 'Left & Right' does not necessarily mean the Sheet Entry is 'I/O'.

Side The Sheet Entry may be positioned on either the left or right hand side of the Sheet Symbol. The Sheet Entry has a fixed size and is positioned within the surrounding border of the Sheet Symbol.

Position The relative position of the Sheet Entry from the top of the Sheet Symbol. The number in this field indicates the grid spacing from the top of the Sheet Symbol. The number of Sheet Entry symbols that can be placed along the vertical sides of the Sheet Symbol will vary with different Sheet Symbol sizes.

Border Color The surrounding border of a Sheet Entry 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. Advanced Schematic will display all the available colors that your graphics card and monitor supports.

Fill Color The inside area of a Sheet Entry 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. Advanced Schematic will display all the available colors that your graphics card and monitor supports.

Text Color Sheet Entry names or labels can be assigned a color. To assign a new color to the Name, click in the color box to open the Color Selector dialog box. Advanced Schematic will display all the available colors that your graphics card and monitor supports.

Selection Toggle the selection state of the Sheet Entry by turning this option on or off. If this option is on, the Sheet Entry will be outlined in the selection color defined in the Options Preferences dialog box.

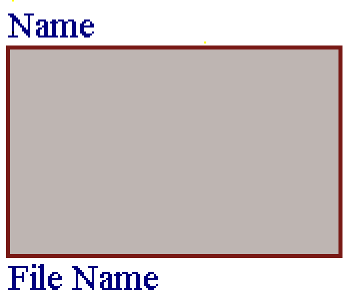
Selections are generally made three ways; using the option in a dialog box, using SHIFT+LEFT MOUSE to add (or remove) individual objects to the current selection and by using the Edit Select and DeSelect commands.

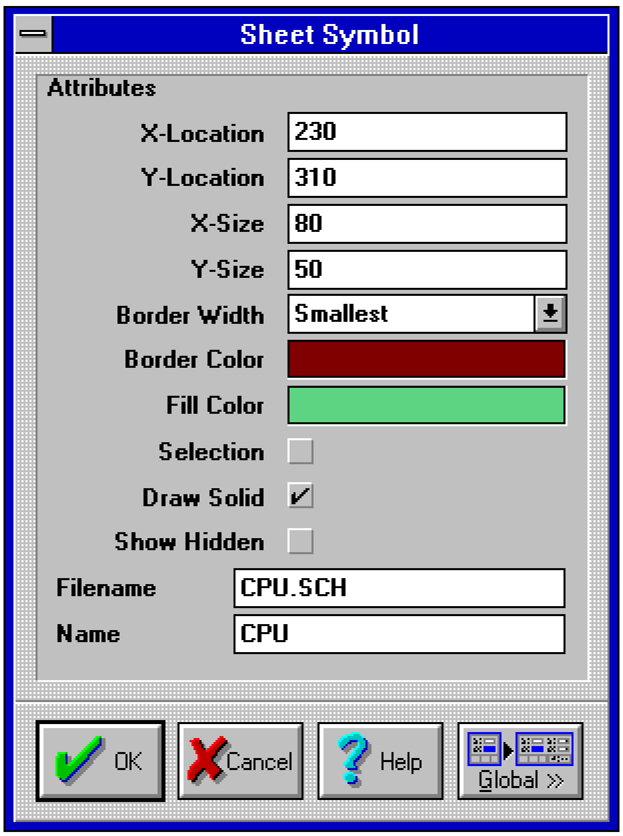
See also

Add Sheet Entry
Sheet Symbol

Sheet Symbol

Overview Sheet symbols represent another schematic sheet in a hierarchical design. Sheet symbols include Sheet Entry symbols, which provide a connection point for signals between the parent and child sheets.





Fields

X-Location The reference point coordinate of a Sheet Symbol, along the x-axis.

Y-Location The reference point coordinate of a Sheet Symbol, along the y-axis.

X-Size Advanced Schematic allows you to change the horizontal width of Sheet Symbols. The width can be any integer (whole number) value. A value of 1 = 10mil = .254mm.

Y-Size Advanced Schematic allows you to change the vertical height of Sheet Symbols. The height can be any integer (whole number) value. A value of 1 = 10mil = .254mm.

Border Width Specifies the border outline thickness of Sheet Symbols. There are four selectable border widths; smallest, small, medium and large.

Border Color The surrounding border of Sheet Symbols can be assigned a color. To assign a new border color, click in the color box to open the Color Selector dialog box. Advanced Schematic will display all the available colors that your graphics card and monitor supports.

Fill Color The inside area of Sheet Symbols 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. Advanced Schematic will display all the available colors that your graphics card and monitor supports.

Selection Toggle the selection state of Sheet Symbols by turning this option on or off. If this option is on, the Sheet Symbol 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 Advanced Schematic allows you to turn the inside area color of Sheet Symbols on or off. If this option is on, the inside area of the Sheet Symbol will be displayed in the fill color. If this option is off, then only the Sheet Symbols border will be shown.

Show Hidden Sheet Symbols have two text fields attached to them, a 'sheet name' and a 'sheet file name', both of these fields can be hidden from the worksheet. If this option is enabled then the hidden names will be displayed. Enabling this option does not change the 'hidden' attribute of the names. The 'hidden' attribute can only be changed by editing the individual sheet name and sheet file name.

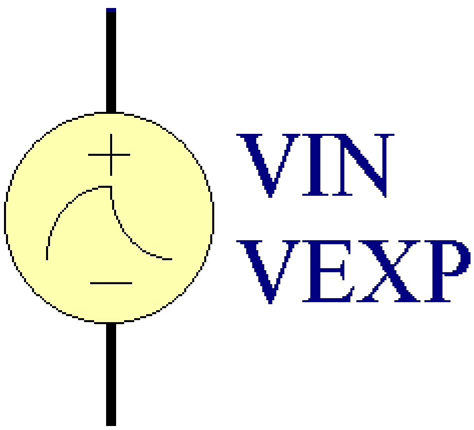
See also

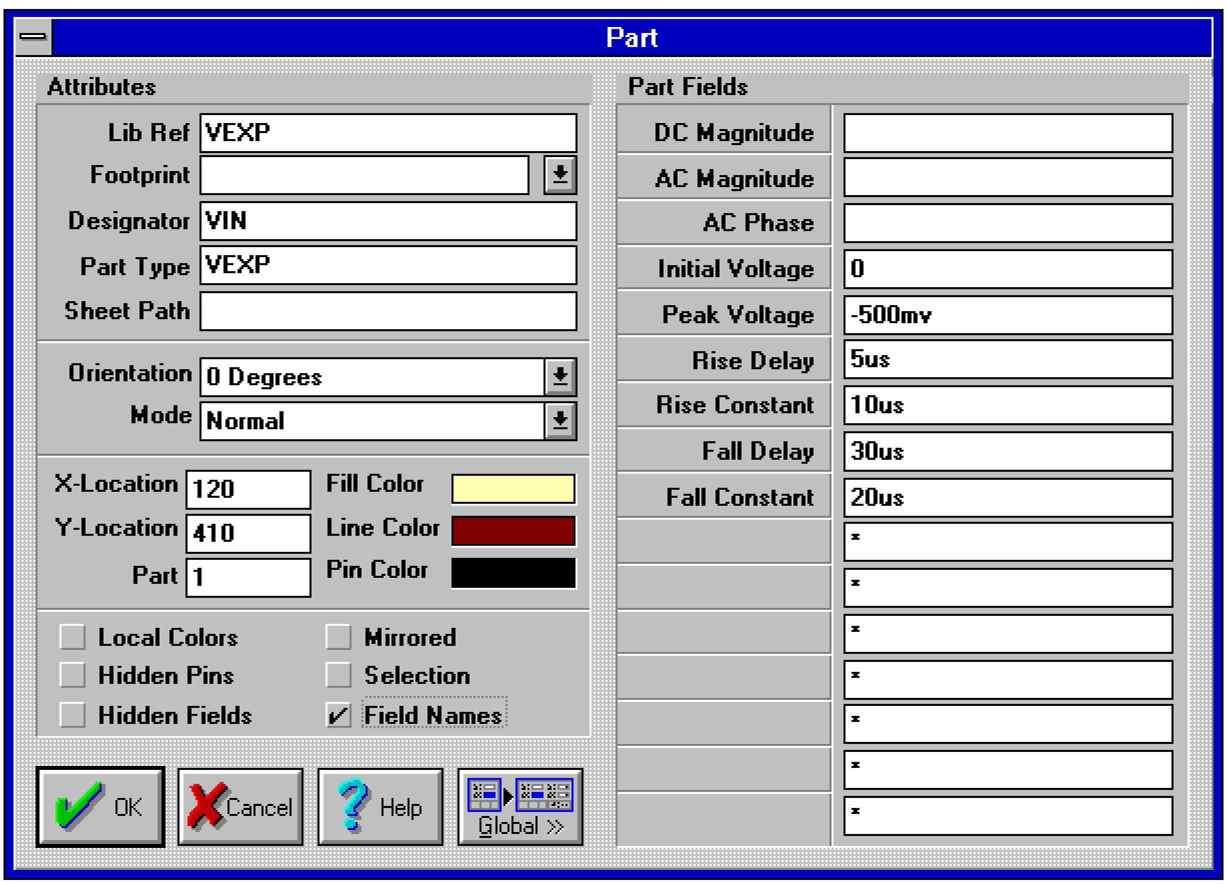
Place Sheet Symbol
Sheet Symbol File Name
Sheet Symbol Name
Sheet Entry
Up Hierarchy
Down Hierarchy

Part

Overview Parts are groups of primitives you place on the schematic worksheet to represent the electronic devices in a design.

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





Fields

X-Location The reference point coordinate of a Part, along the x-axis.

Y-Location The reference point coordinate of a Part, along the y-axis.

Mode Parts can be displayed in three different modes, Normal, De-Morgan and IEEE.

Fill Color The inside area of Parts 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. Advanced Schematic will display all the available colors that your graphics card and monitor supports. This color will only be used if the local color field is enabled.

Line Color The surrounding border of Parts 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. Advanced Schematic will display all the available colors that your graphics card and monitor supports. This color will only be used if the local color field is enabled.

Pin 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. Advanced Schematic will display all the available colors that your graphics card and monitor supports. This color will only be used if the local color field is enabled.

Local Colors When this option is enabled, the default colors of the Part defined in the Library Editor will be over ridden by the Fill color, Line color and Pin color defined in this dialog box.

Mirrored When this option is enabled, the Part is flipped along its x-axis. While moving a part, you can flip it along its x or y axis by pressing the X and Y keys.

Orientation Parts can be rotated 360 degrees using 90 degree increments. There are four selectable orientations, 0 degrees, 90 degrees, 180 degrees and 270 degrees. While moving a part, you can rotate it around the cursor by pressing the SPACEBAR.

Selection Toggle the selection state of Parts by turning this option on or off. If this option is enabled, the Part 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.

Part This field signifies the part number of the component, such as U5:A or U5:1 signifies the first part of the component. If the component has multiple parts then changing this value will update the components part. You can also change the components part by choosing Increment Part Number from the Edit menu and then clicking on the component to toggle through the available parts.

Hidden Fields Parts have several attached fields, a Part Type, Part Designator, and eight text description fields, any one of these fields can be hidden from the worksheet. If this option is enabled then the hidden names will be displayed. Enabling this option does not change the 'hidden' attribute of the field names. The 'hidden' attribute can only be changed by editing the individual names.

Hidden Pins If this option is enabled then all the hidden pins on the part will be shown. Power pins are commonly defined as being hidden, such as +5, VCC, -12, GND, VSS etc. Enabling this option does not change the 'hidden' attribute of pins. The 'hidden' attribute can only be change in the Library Editor.

Lib Ref Specifies the Parts' library paSheet symbols represent another schematic sheet in a hierarchical design. Sheet symbols include Sheet Entry symbols, which provide a connection point for signals between the parent and child sheets.

Comments Global editing of parts is performed at a more sophisticated level compared to the ordinary global editing of other objects. The "Copy To" fields allows direct reference to all the text attributes of a part. This provides a powerful text manipulation capability that can be used to copy any combination of text fields to any other fields within a part. Please refer to the "New Features" section for a detailed description of this feature.

See also

Place Parts
Part Description
Part Type
Part Designator
Sheet Part File Name

MainMenu

File	Nested Menu
Edit	Nested Menu
Place	Nested Menu
Library	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, empty schematic worksheet
Open Sheet...	Open and load a schematic worksheet file
Open Project...	Open and load all schematic sheets in a project
Close	Close the current window
Close Project	Close all open schematic windows
Save	Save current schematic worksheet with same file name
Save As...	Save current schematic worksheet with new file name
Save All	Save all loaded schematic project and worksheet files
Save Project	Save all the sheets in the current project
Setup Printer...	Printer and page setup
Print	Print the current document with the last set printer options
Annotate...	Re-Designate all Parts in schematic project
Back Annotate...	Back Annotate Part and Pin information from a PCB Was/Is File
Create Netlist...	Generate a netlist from the current project or sheet
Hierarchy	Nested Menu
Reports	Nested Menu
Exit	Quit from Advanced Schematic

Hierarchy

Down Hierarchy	Switch to a child sheet of the current sheet
Up Hierarchy	Switch to the parent sheet of the current sheet
Complex To Simple	Convert Complex Hierachical Design To Simple Hierarchical Design
Create Sheet From Symbol	Creates a new sheet (File New command) and adds ports for sheet entries on the sheet symbol
Create Symbol From Sheet	Create a sheet symbol that represents the current sheet

Reports

Bill of Material...	Generate a Bill of Materials
Project Hierarchy...	Generate a text listing to represent the design hierarchy
Cross Reference...	Generate a part/sheet cross reference for the current project or sheet
Electrical Rules Check...	Perform an Electrical Rules Check on the current project or sheet
Netlist Compare...	Compare two Protel format netlists and generate a report

Edit

Undo	Undo previous command
Redo	Redo previous undo command
Cut	Copy selected objects to clipboard and remove from sheet
Copy	Copy all selected objects to the clipboard
Paste	Place clipboard contents onto current worksheet
Paste Array...	Define an array placement of clipboard contents
Clear	Delete all selected objects from the current worksheet
Find Text...	Search for and jump to a text string on any sheet
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 schematic worksheet
Change	Select and use dialog to change objects
Move	Nested Menu
Align	Nested Menu
Jump	Nested Menu
Increment Part Number	Toggle part number on multi-part component
Cross Probe Part On PCB	Select part to cross probe to PCB
Cross Probe Pin On PCB	Select pin to cross probe to PCB
Cross Probe Net On PCB	Select power object / Net Label to cross probe to PCB

Select

Inside Area	Select all objects inside an area
Outside Area	Select all objects outside an area
All	Select everything on the current worksheet
Net	Find physical connection from a point
Connection	Find physical connection between pins

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

Drag	Select and move objects together with connected wires/buses
Move	Select and move objects on schematic sheet
Move Selection	Move selected objects to another area of the worksheet
Drag Selection	Move selected object(s) and drag connected objects
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

Jump To Error Marker	Jump to the next Error Marker in the Project
Origin	Jump to the origin of the worksheet (lower left)
New Location...	Type in and jump to a new location on the worksheet
Location Mark 1	Jump to a location previously specified by setting Location Mark 1-10.
Location Mark 2	Jump to a location previously specified by setting Location Mark 1-10.
Location Mark 3	Jump to a location previously specified by setting Location Mark 1-10.
Location Mark 4	Jump to a location previously specified by setting Location Mark 1-10.
Location Mark 5	Jump to a location previously specified by setting Location Mark 1-10.
Location Mark 6	Jump to a location previously specified by setting Location Mark 1-10.
Location Mark 7	Jump to a location previously specified by setting Location Mark 1-10.
Location Mark 8	Jump to a location previously specified by setting Location Mark 1-10.
Location Mark 9	Jump to a location previously specified by setting Location Mark 1-10.
Location Mark 10	Jump to a location previously specified by setting Location Mark 1-10.

Place

Bus	Place bus lines on the worksheet
Bus Entry	Place bus entry on the worksheet
Part	Place parts on the worksheet
Junction	Place a junction on the current worksheet
Power Port	Place power ports on the worksheet
Wire	Place electrical wires on the worksheet
Net Label	Place a net label on the worksheet
Port	Place a port on the sheet
Sheet Symbol	Place hierarchical sheet symbols on the worksheet
Add Sheet Entry	Add net connections to sheet symbols
Drawing Tools	Nested Menu
Text	Nested Menu
Directives	Nested Menu
Location Marks	Nested Menu

Drawing Tools

Arcs	Place graphical arcs on the worksheet
Elliptical Arcs	Place elliptical arcs on the worksheet
Ellipses	Place elliptical shapes on the sheet
Pie Charts	Place pie shapes on the sheet
Line	Place graphical lines on the sheet
Rectangle	Place rectangles on the worksheet
Round Rectangle	Place round rectangles on the worksheet
Polygons	Place graphical polygon shapes on the worksheet
Beziers	Place bezier curves on the sheet
Graphic...	Place PCX/BMP/GIF/TIFF/WMF/EPS graphical images on the sheet

Text

Text Frame	Place multi-line text frames on the sheet
Annotation	Place single line text on the sheet

Directives

No ERC	Suppress no-connection error message for nets
Probe	Add net to list of nets to be traced by digital simulator
Test Vector Index	Mark nets to identify which simulation vector to use
Stimulus	Add stimulus information to nets for digital simulation
PCB Layout	Add directives to nets for PCB routing

Location Marks

Set Location Mark 1	Set location mark (1-10) on the current worksheet
Set Location Mark 2	Set location mark (1-10) on the current worksheet
Set Location Mark 3	Set location mark (1-10) on the current worksheet
Set Location Mark 4	Set location mark (1-10) on the current worksheet
Set Location Mark 5	Set location mark (1-10) on the current worksheet
Set Location Mark 6	Set location mark (1-10) on the current worksheet
Set Location Mark 7	Set location mark (1-10) on the current worksheet
Set Location Mark 8	Set location mark (1-10) on the current worksheet
Set Location Mark 9	Set location mark (1-10) on the current worksheet
Set Location Mark 10	Set location mark (1-10) on the current worksheet

Library

Add/Remove...	Add and remove libraries from the library list
Run Library Editor...	Switch to the Schematic Library Editor
Make Project Library...	Make a library of parts contained in the current project
Update Parts In Cache	Update all parts from library information
Find Component...	Search for a component in a specified path.

Tools

Schematic Library Editor	Switch to the Schematic Library Editor
Analog Simulator	Launch the Analog Simulator spcified in the Tools-Options Dialog
Digital Simulator	Launch the Digital Simulator spcified in the Tools-Options Dialog
Mixed-Signal Simulator	Launch the Mixed-Signal Simulator spcified in the Tools-Options Dialog
PCB Layout Editor	Switch to the PCB Layout Editor
PLD/FPGA Compiler	Launch the PLD Compiler spcified in the Tools-Options Dialog
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 PBRUSH.EXE)
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
Sheet...	Setup options for current worksheet
Hot Keys...	Setup Hot Key assignments
Auto-Pan...	Setup Autopan options
Memory Monitor...	Setup low memory/resources warning thresholds.
Update Current Template	Update the current sheet from its template file.
Set Template File Name...	Change the current template to a different template file
Remove Template	Remove any template information from the current sheet
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 Toolbar	Turn the Main Toolbar on or off
Wiring Toolbar	Turn the Wiring Toolbar on or off
Drawing Toolbar	Turn the Drawing Toolbar on or off
Project Manager	Turn the Project Manager on or off
Component Browser	Turn the Component Browser on or off
Visible Grid	Turn the visible grid on or off
Snap Grid	Turn the snap grid on or off
Electrical Grid	Turn the electrical grid on or off
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 sheet and fit that area in the window
Point	Select a rectangular area of the sheet 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 worksheet (higher magnification)
Out	Show more of the current worksheet (lower magnification)
Pan	Re-Center the screen around the cursor
Redraw	Update the screen display
All	Fit all objects on the current sheet in the window
Sheet	Show entire sheet

Info

System Status...	Display system information
Selected Pins...	List all selected pins

Window

Tile	Tile all open schematic windows
Cascade	Cascade all open schematic windows
Arrange Icons	Arrange all minimized open schematic windows
Close All	Close all open schematic windows

Help

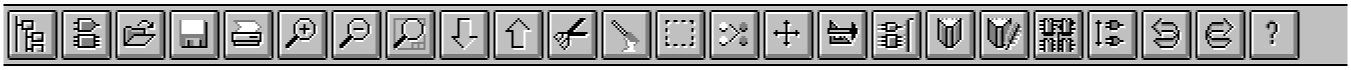
New Features	Information on new features
Contents	Help system topic index
Using Help	Information about the Windows Help System
Basic Concepts	Basic information about Advanced Schematic
Commands	Help information organized by menu structure
Printing	Information about generating hard copy output
Reference	Advanced Schematic Reference
DOS Schematic	Command Cross Reference for Protel DOS Schematic users
Orcad SDT	Command Cross Reference for OrCAD SDT users
About...	Display the version number and copyright of Advanced Schematic

Hot Keys

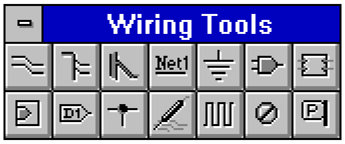
M	**Popup Move Menu** Popup the Move Menu
E	**Popup Edit Menu** Popup the Edit menu
P	**Popup Place Menu** Popup the Place menu
J	**Popup Jump Menu** Popup the Edit Jump menu
S	**Popup Select Menu** Popup the Edit Select menu
X	**Popup De-Select Menu** Popup the Edit De-Select menu
Z	**Popup Zoom Menu** Popup the Zoom menu
O	**Popup Options Menu** Popup the Options menu
H	**Popup Help Menu** Popup the Help menu
F	**Popup File Menu** Popup the File menu
L	**Popup Library Menu** Popup the Library menu
T	**Popup Tools Menu** Popup the Tools menu
A	**Popup Alignment Menu** Popup the Alignment 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 sheet
Ctrl Insert	**Copy** Copy all selected objects to the clipboard
Shift Insert	**Paste** Place clipboard contents onto current worksheet
Ctrl Delete	**Clear** Delete all selected objects from the current worksheet
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 schematic sheet
MouseDblLeft	**Change Single Object** Use dialog to change object under the cursor
Ctrl MouseLeft	**Drag Single Item** Focus on and move a single object (and drag connected wires)
Ctrl MouseLeftHold	**Drag Single Item** Focus on and move a single object (and drag connected wires)
Ctrl Shift MouseLeft	**Move Single Object** Select and move objects on the schematic worksheet
MouseLeftHold	**Change Object Graphically or Move** Select and change objects on schematic sheet
Ctrl PgDn	**Zoom All** Fit all objects on the current sheet in the window
PgUp	**Zoom In** Show less of the current worksheet (higher magnification)
PgDn	**Zoom Out** Show more of the current worksheet (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 any sheet
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

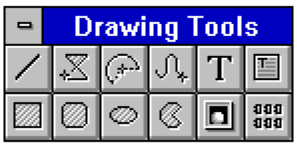
Tool Bars



Tools

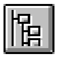


Wiring Tools

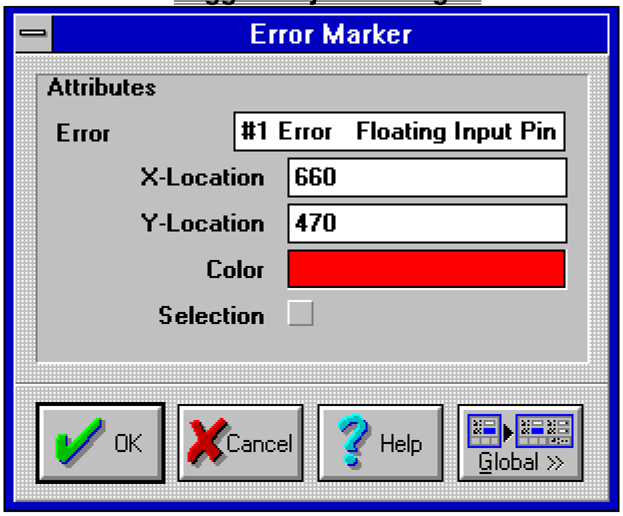


Drawing Tools

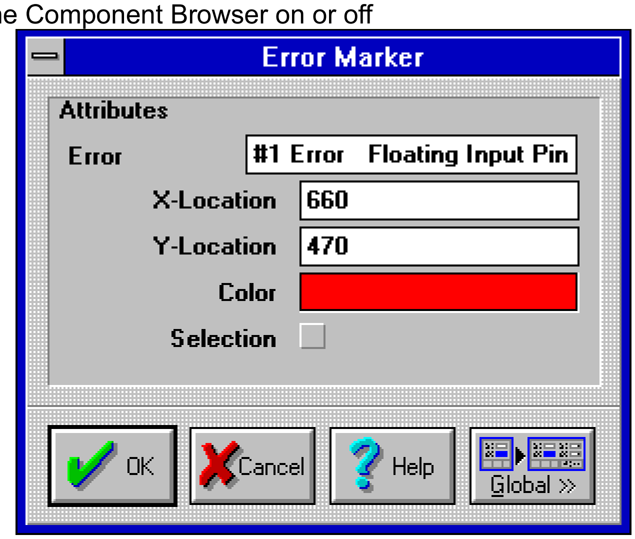
Tools



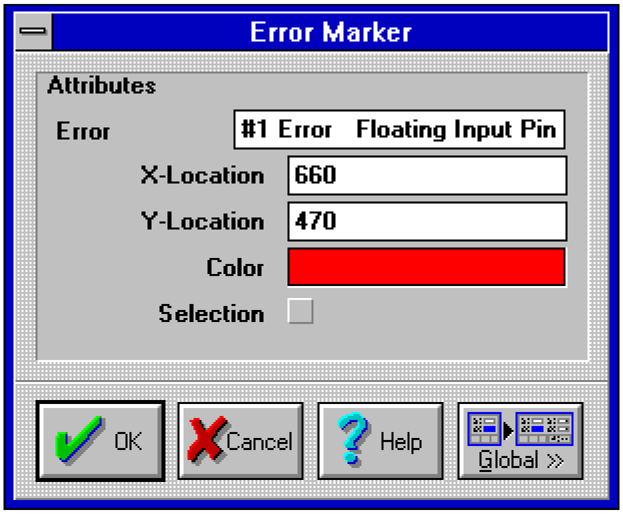
Toggle Project Manager Turn the Project Manager on or off



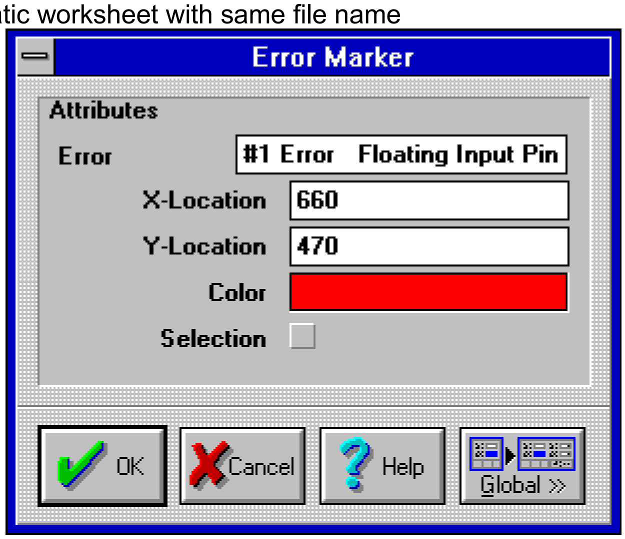
Toggle Component Browser Turn the Component Browser on or off



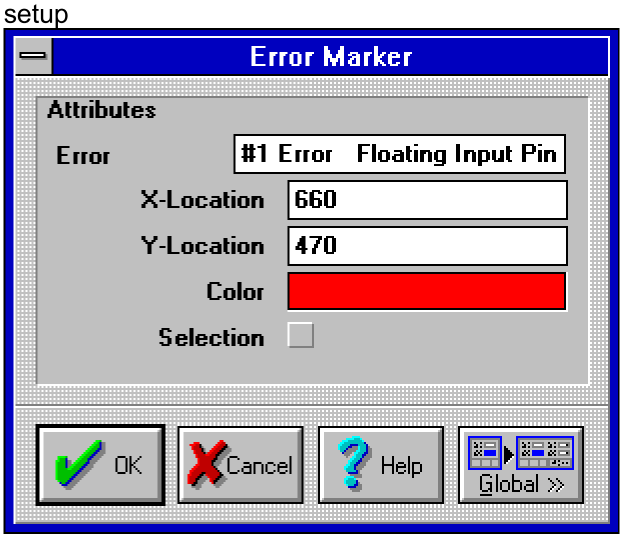
File Open Sheet Open and load a schematic worksheet file



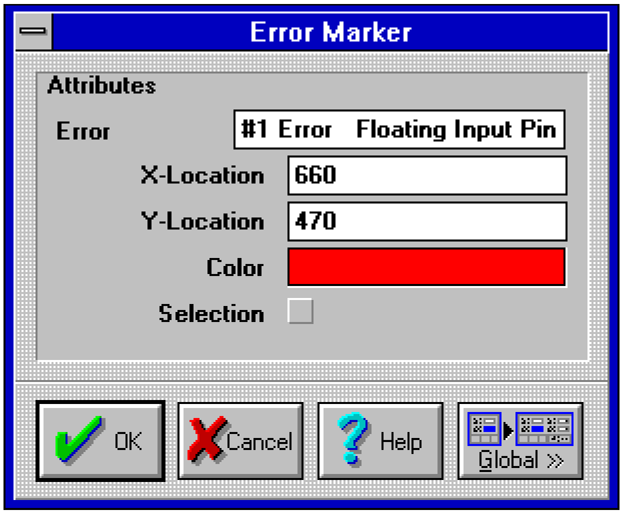
File Save Save current schematic worksheet with same file name



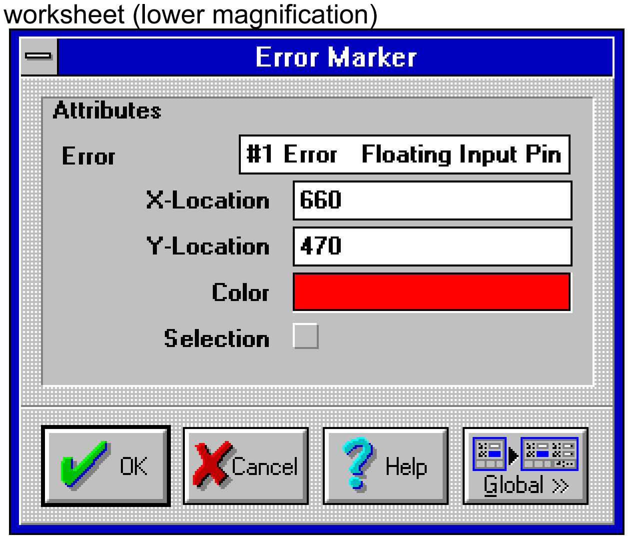
Setup Printer Printer and page setup



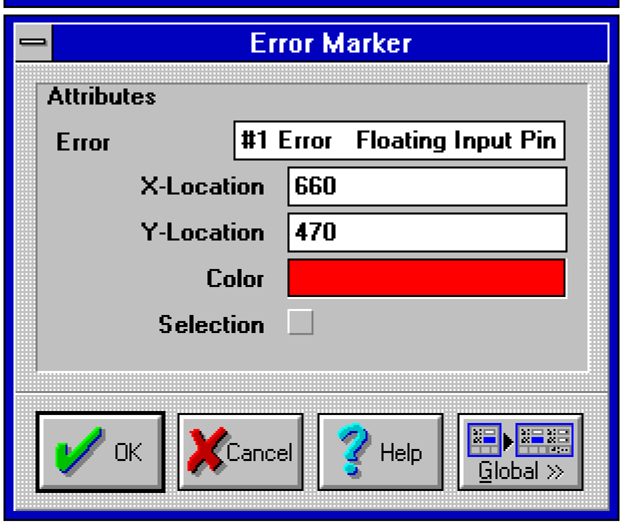
Zoom In Show less of the current worksheet (higher magnification)



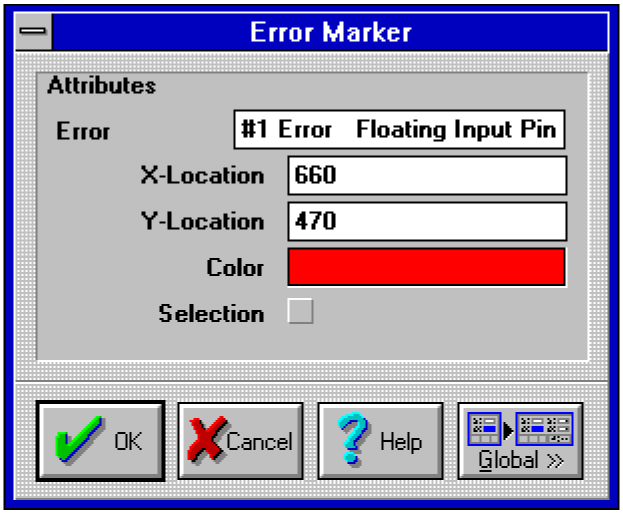
__Zoom Out__ Show more of the current worksheet (lower magnification)



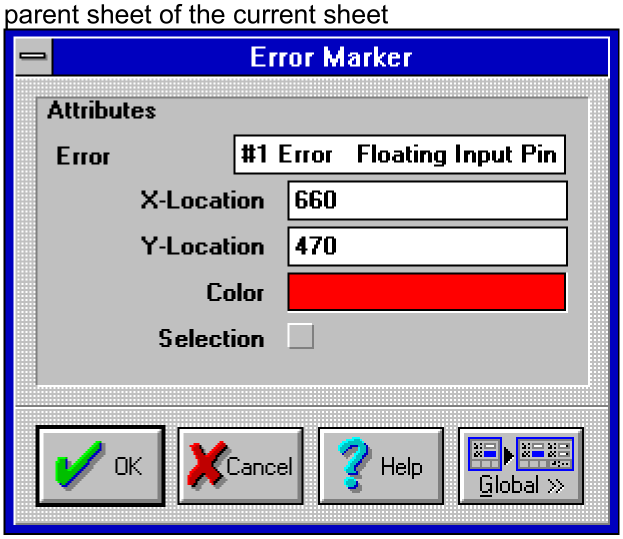
__Zoom Sheet__ Show entire sheet



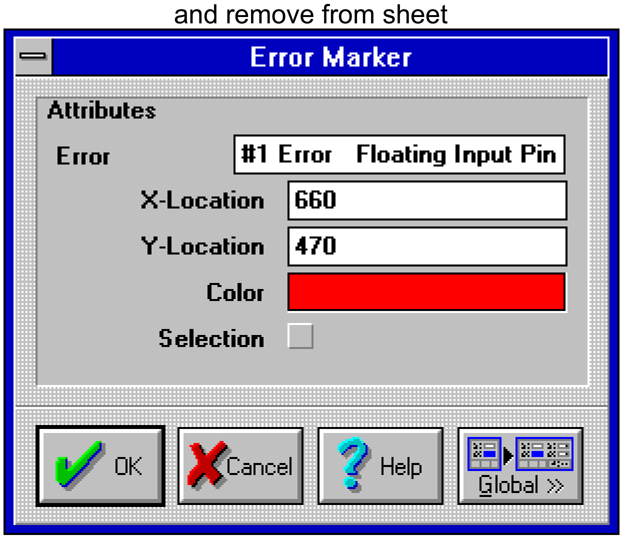
__Down Hierarchy__ Switch to a child sheet of the current sheet



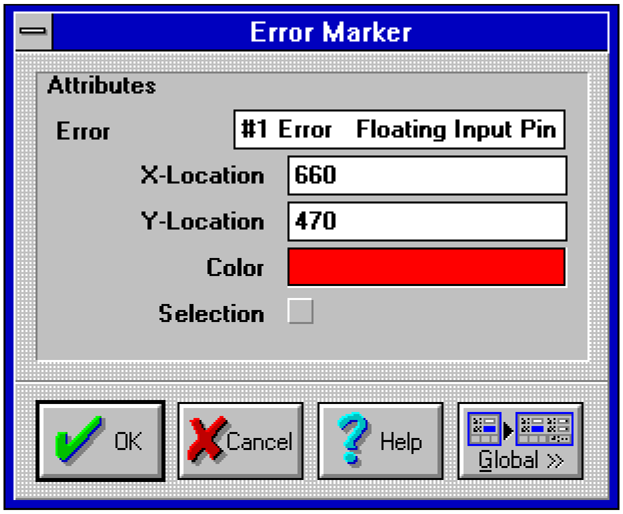
Up Hierarchy Switch to the parent sheet of the current sheet



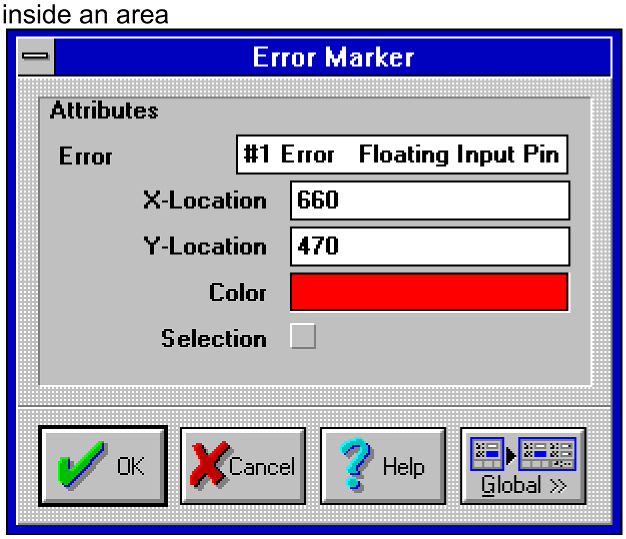
Cut Copy selected objects to clipboard and remove from sheet



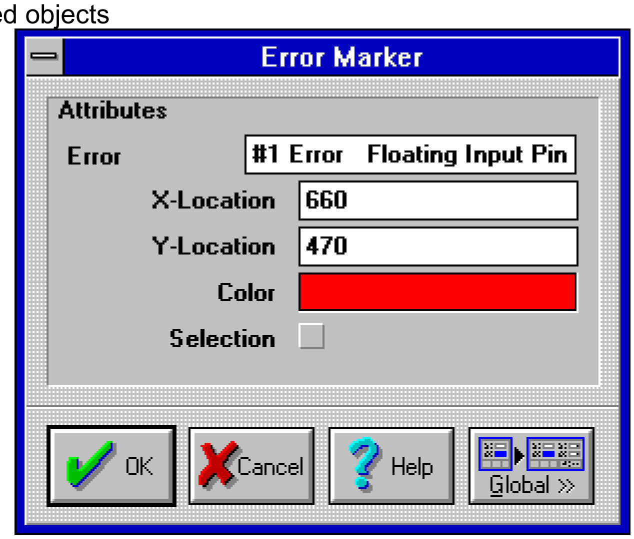
Paste Place clipboard contents onto current worksheet



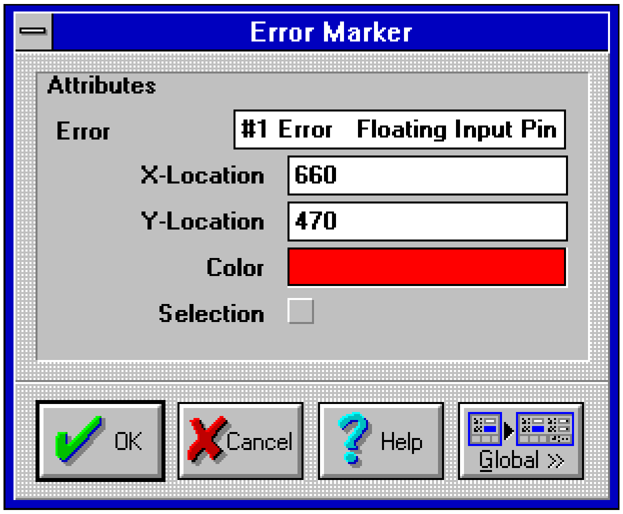
Select Inside Area Select all objects inside an area



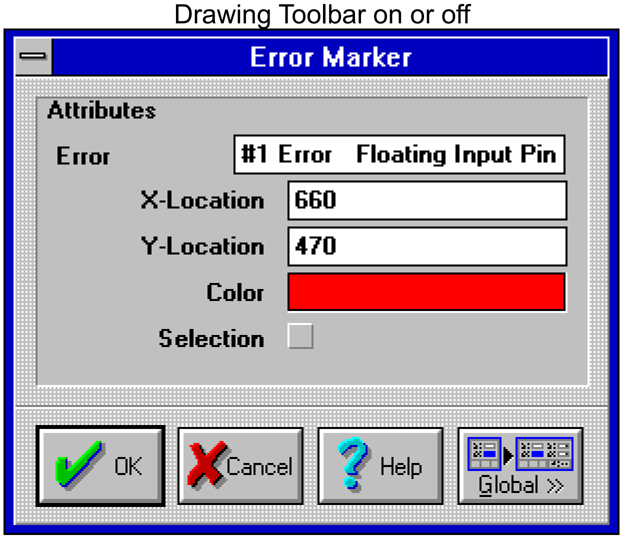
De-Select All De-Select all selected objects



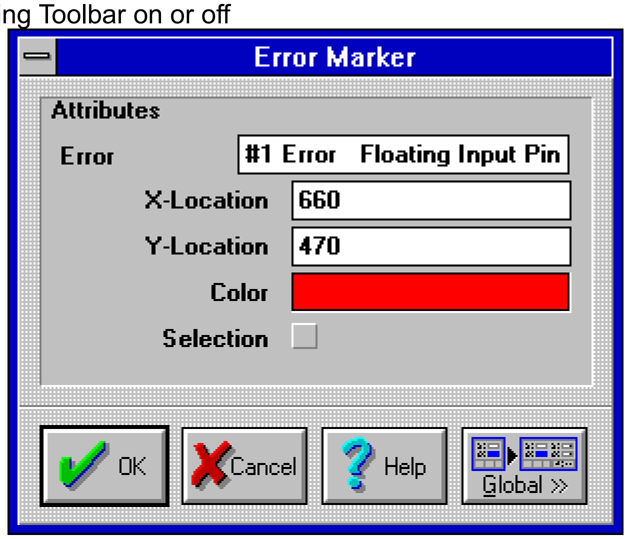
Move Selection Move selected objects to another area of the worksheet



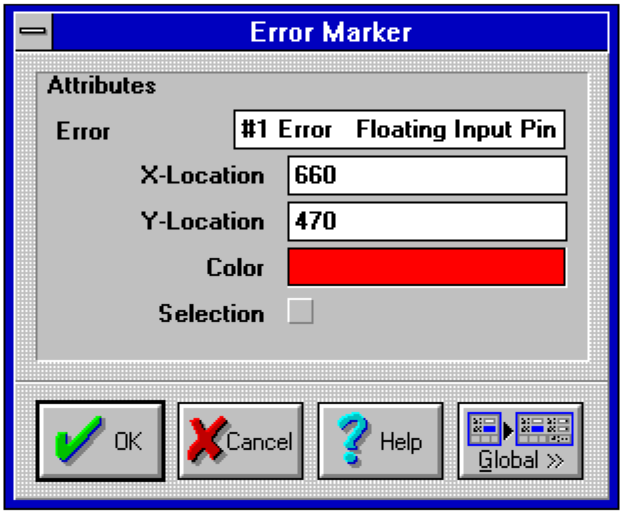
Toggle Drawing Toolbar Turn the Drawing Toolbar on or off



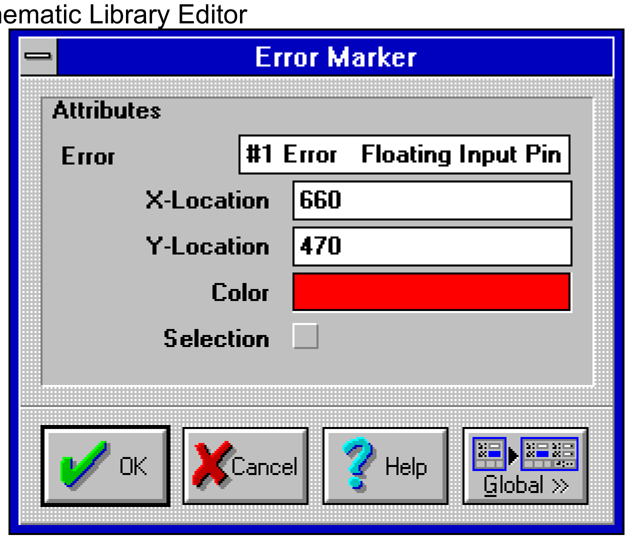
Toggle Wiring Toolbar Turn the Wiring Toolbar on or off



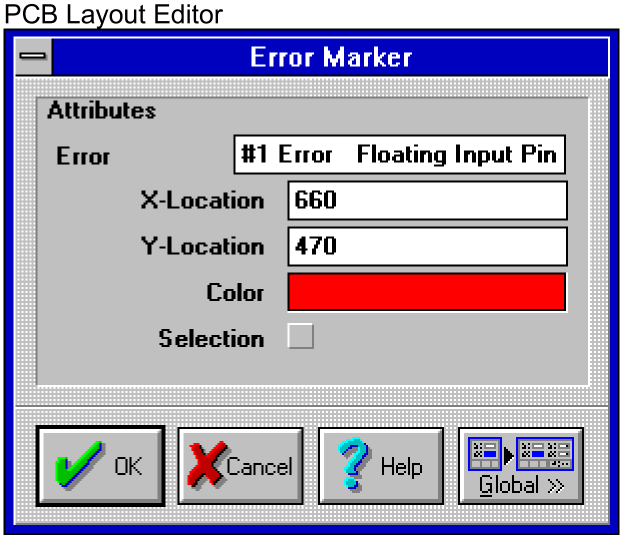
Add/Remove Library Add and remove libraries from the library list



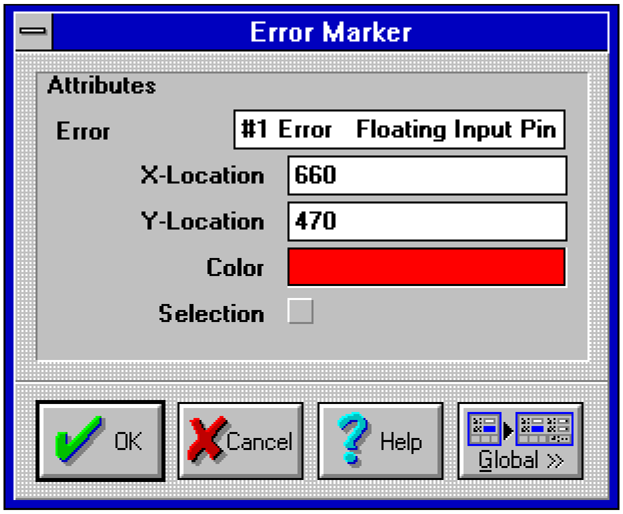
Run Library Editor Switch to the Schematic Library Editor



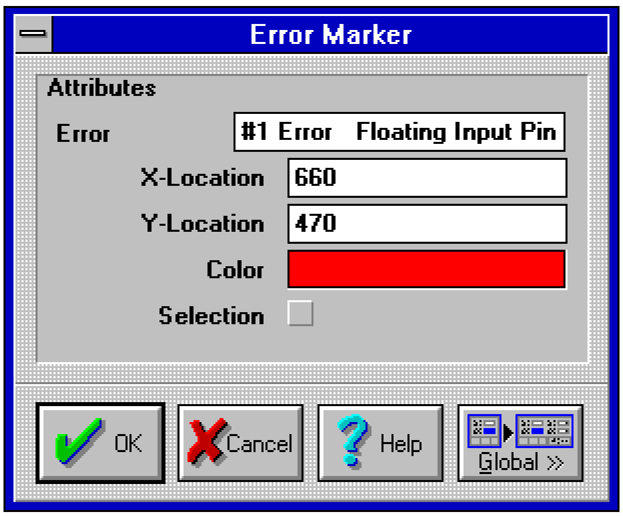
Run PCB Editor Switch to the PCB Layout Editor



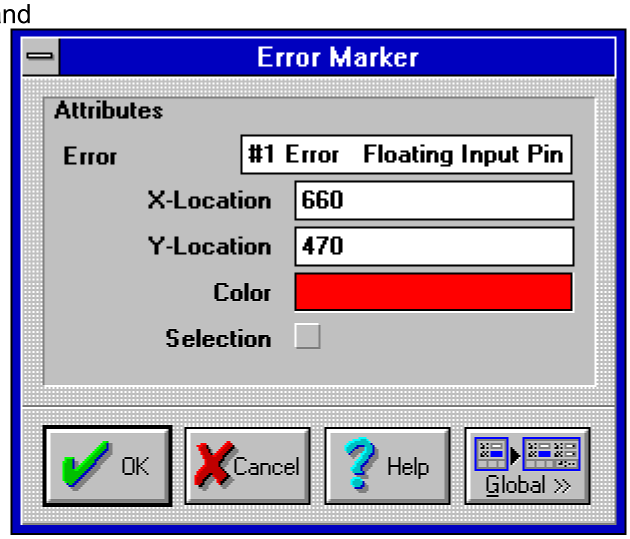
Increment Part Number Toggle part number on multi-part component



Undo Undo previous command

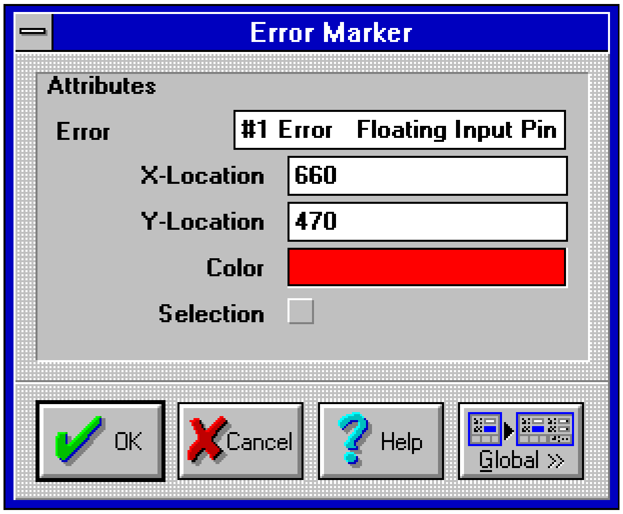


Redo Redo previous undo command

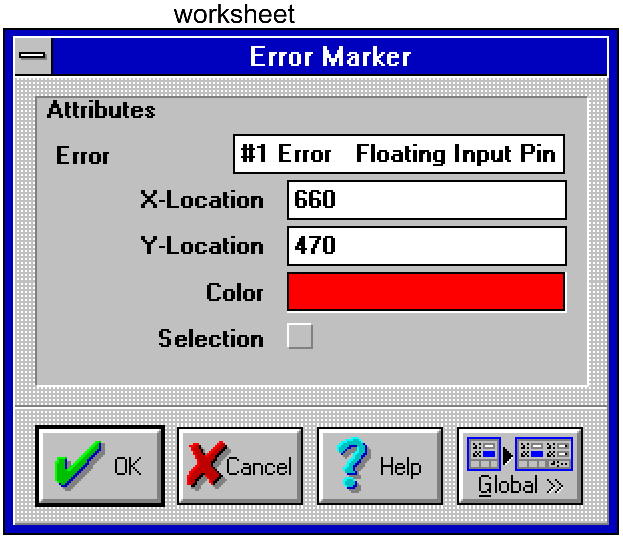


Help Index Help system topic index

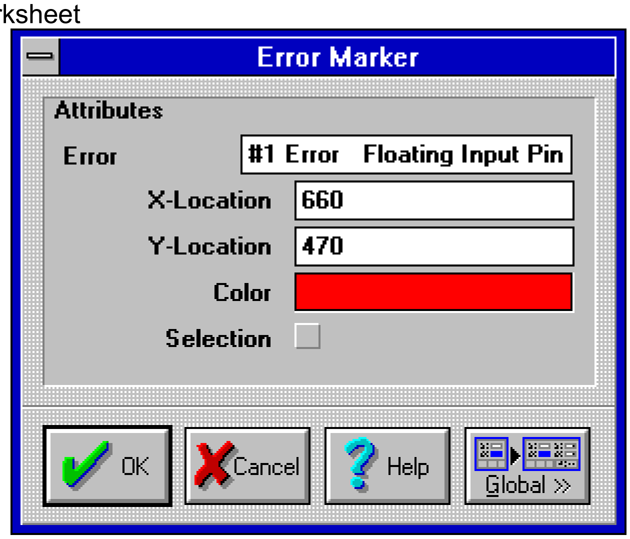
Wiring Tools



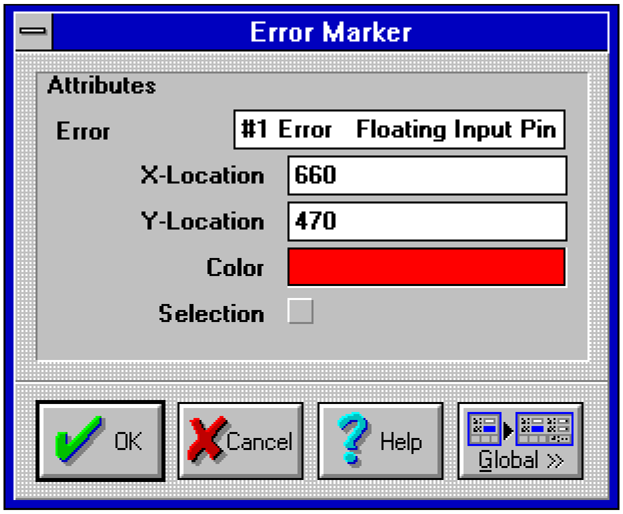
Place Wire Place electrical wires on the worksheet



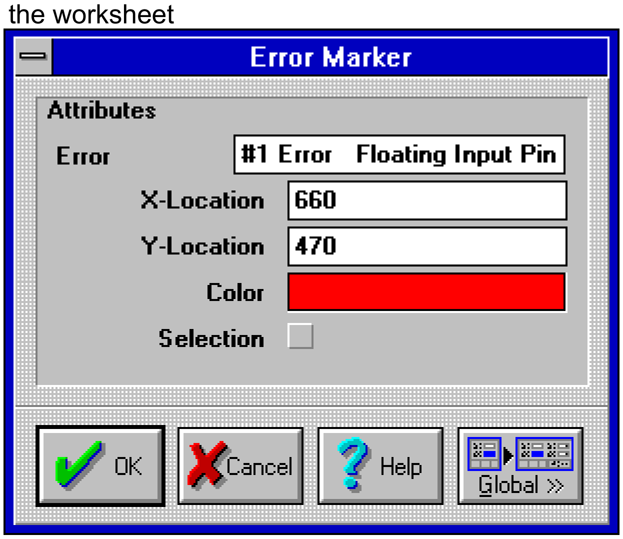
Place Bus Place bus lines on the worksheet



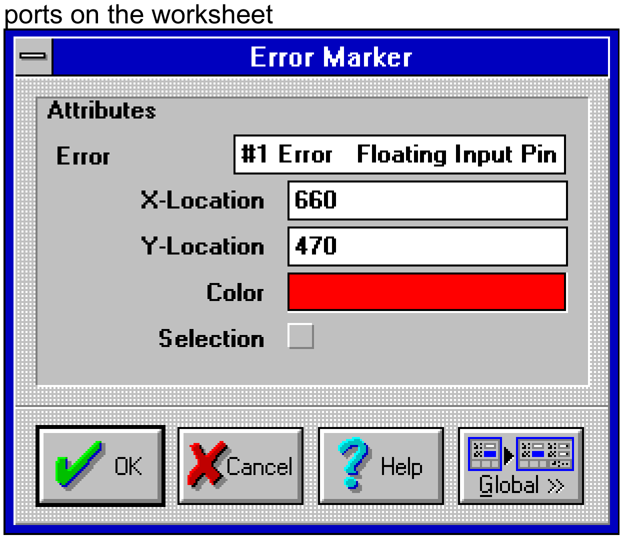
Place Bus Entry Place bus entry on the worksheet



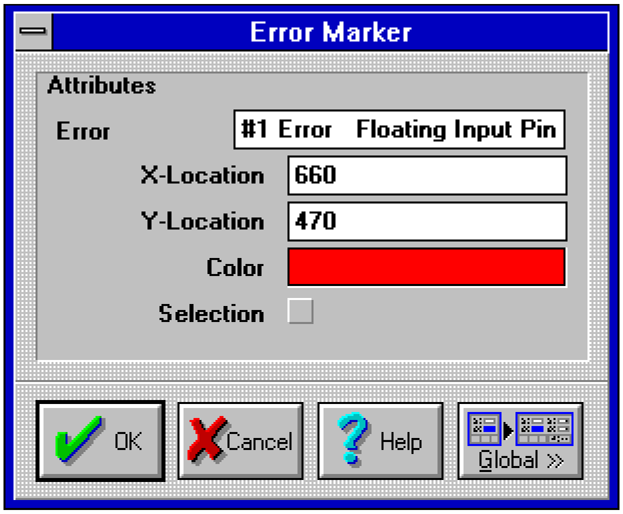
<u>Place Net Label</u> Place a net label on the worksheet



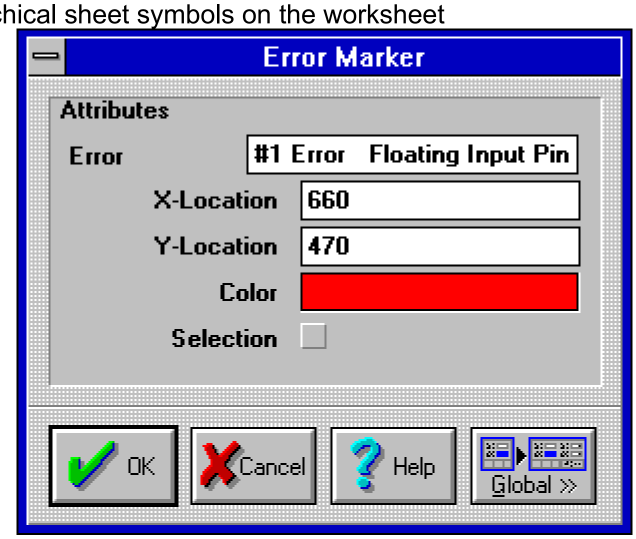
<u>Place Power Port</u> Place power ports on the worksheet



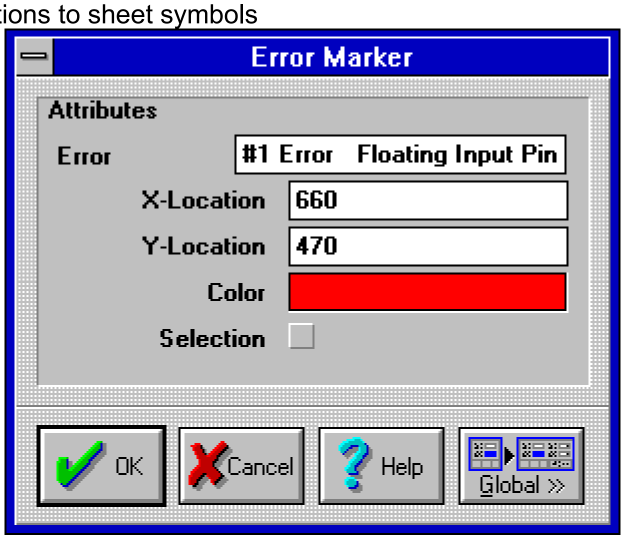
<u>Place Parts</u> Place parts on the worksheet



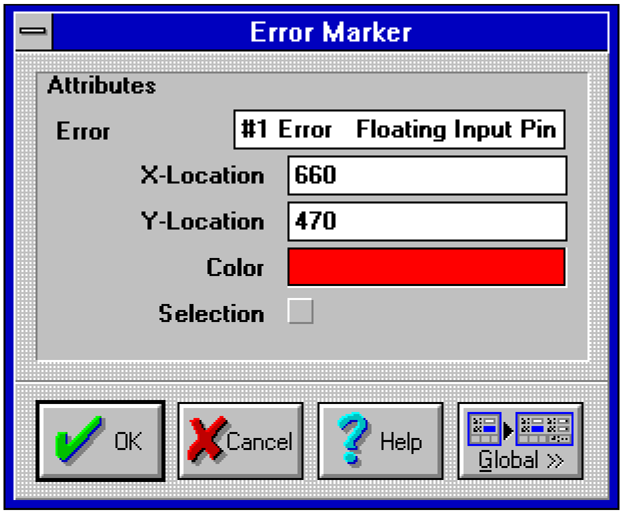
Place Sheet Symbol Place hierarchical sheet symbols on the worksheet



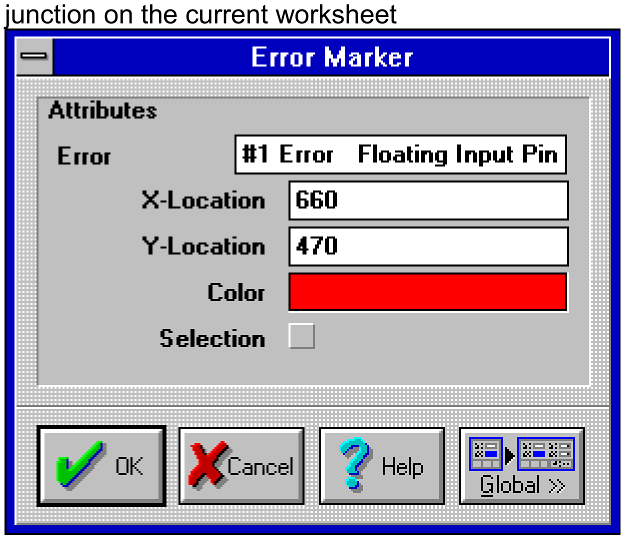
Add Sheet Entry Add net connections to sheet symbols



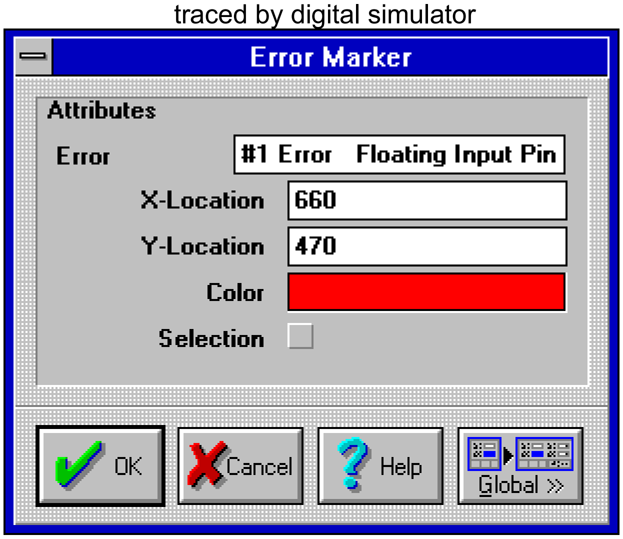
Place Port Place a port on the sheet



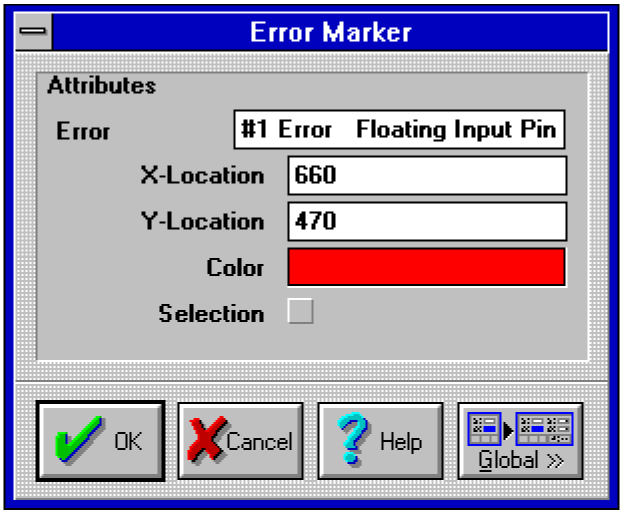
Place Junctions Place a junction on the current worksheet



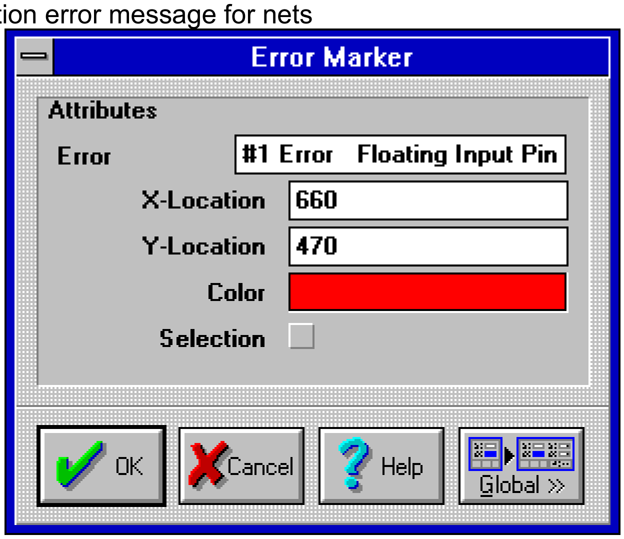
Place Probe Add net to list of nets to be traced by digital simulator



Place Stimulus Add stimulus information to nets for digital simulation

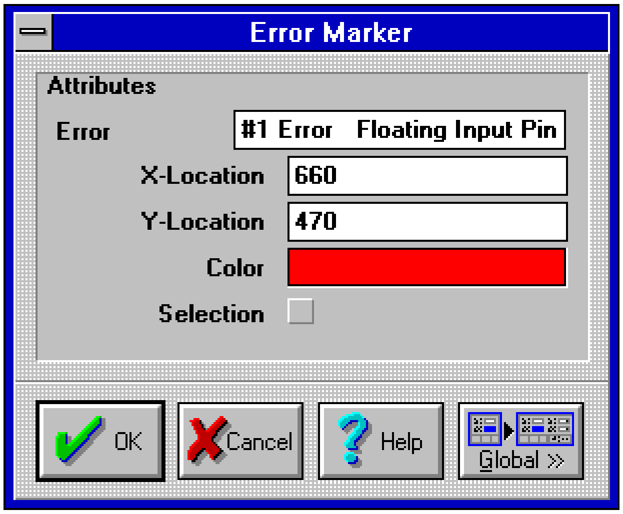


Place No ERC Suppress no-connection error message for nets

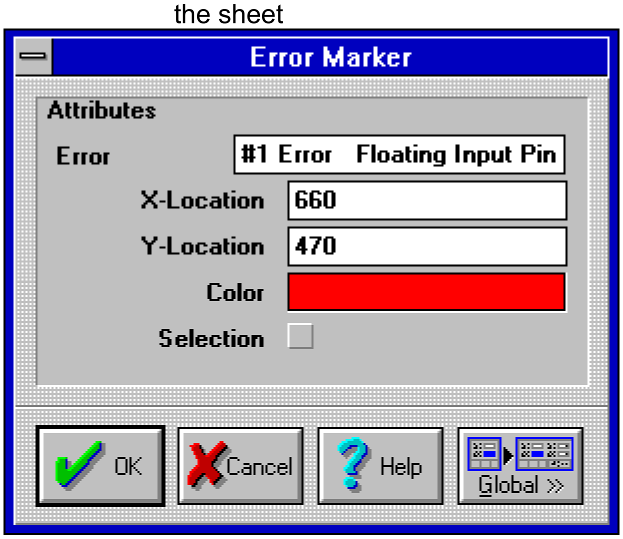


Place PCB Layout Add directives to nets for PCB routing

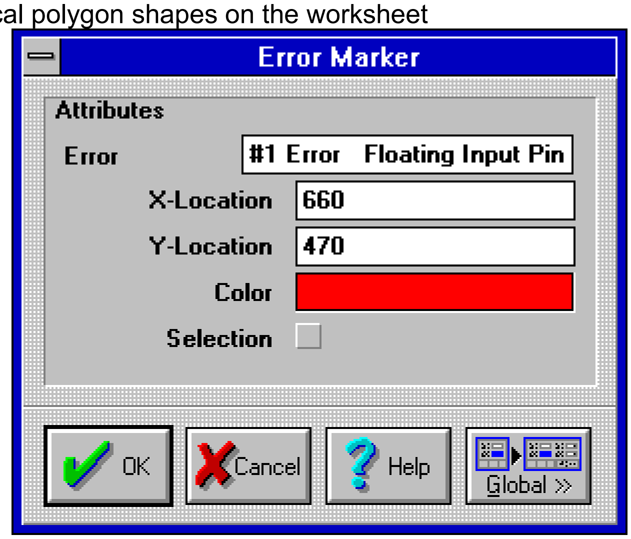
Drawing Tools



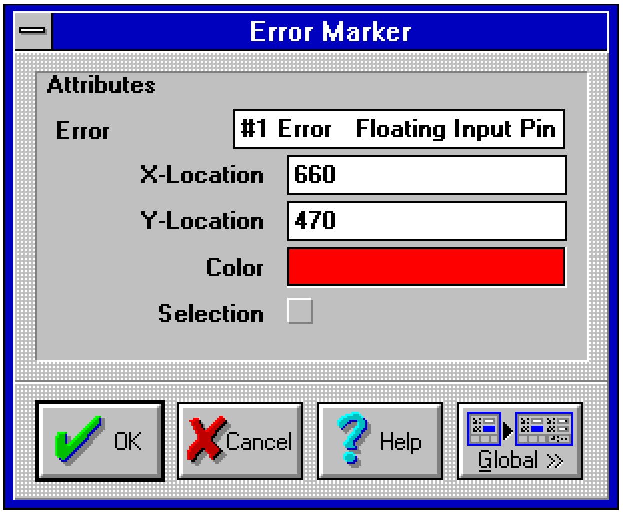
Place Lines Place graphical lines on the sheet



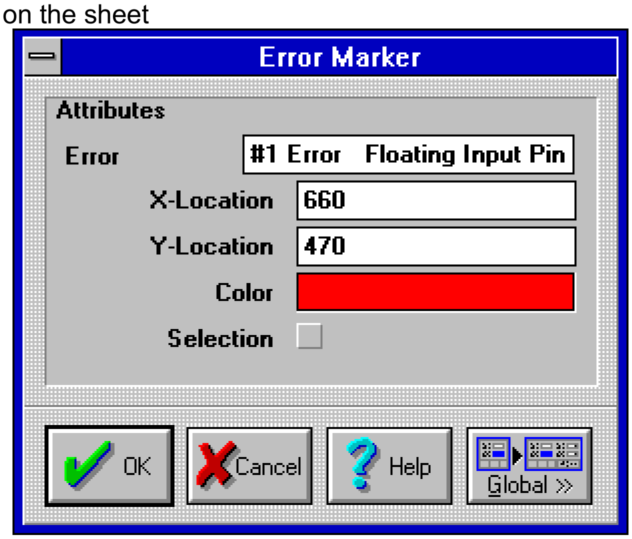
Place Polygons Place graphical polygon shapes on the worksheet



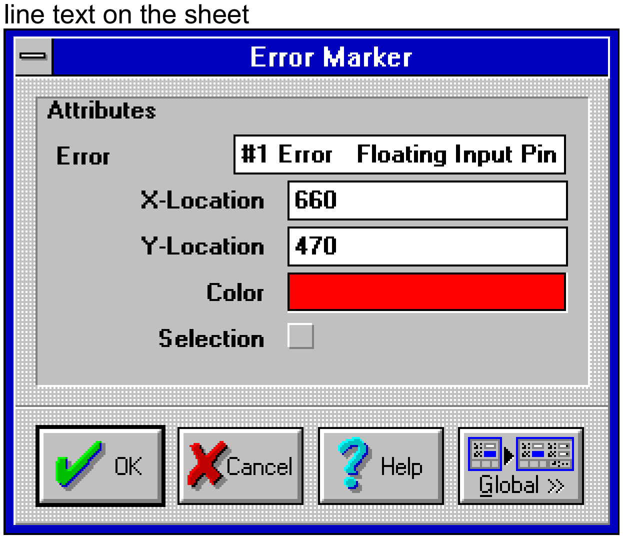
Place Elliptical Arcs Place elliptical arcs on the worksheet



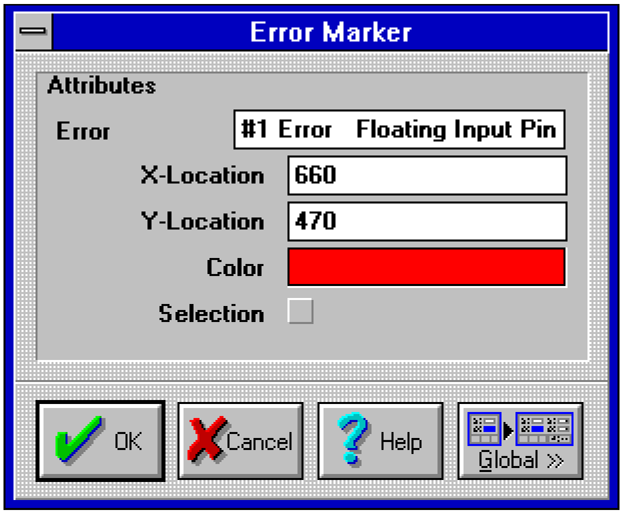
Place Beziers Place bezier curves on the sheet



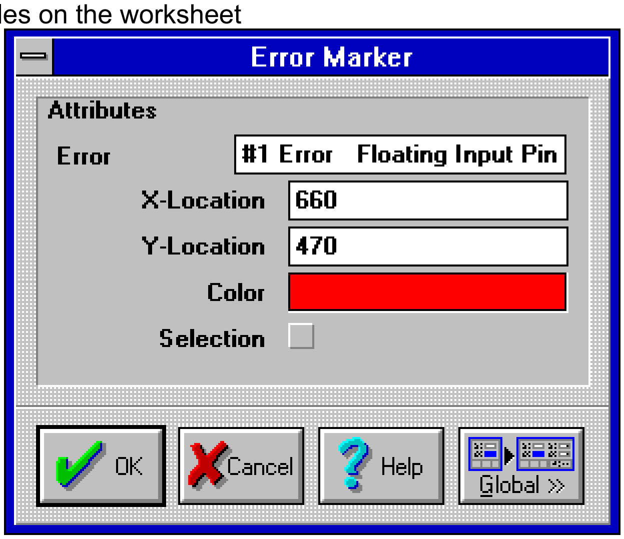
Place Annotation Place single line text on the sheet



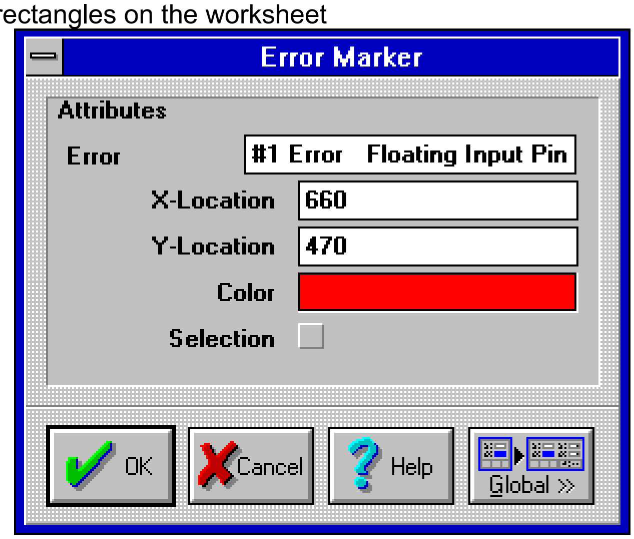
Place Text Frame Place multi-line text frames on the sheet



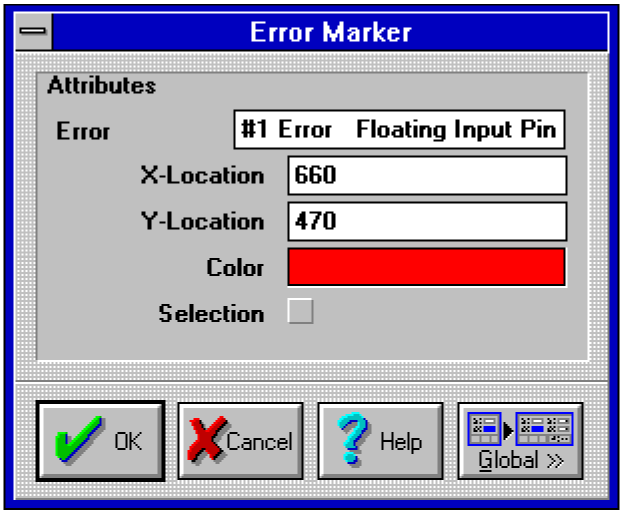
Place Rectangles Place rectangles on the worksheet



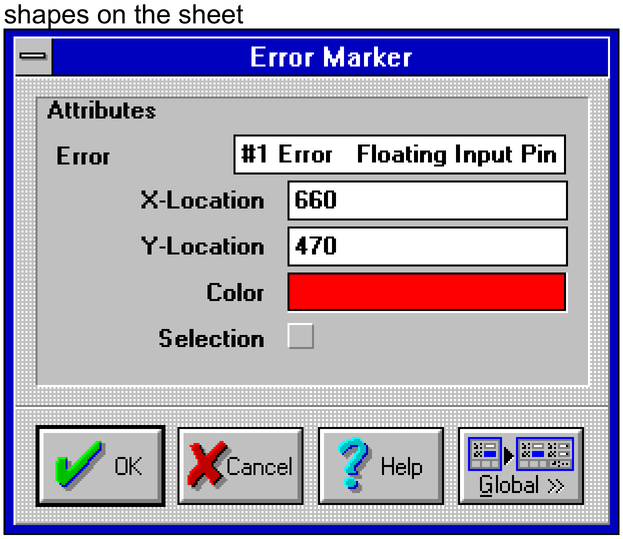
Place Round Rectangle Place round rectangles on the worksheet



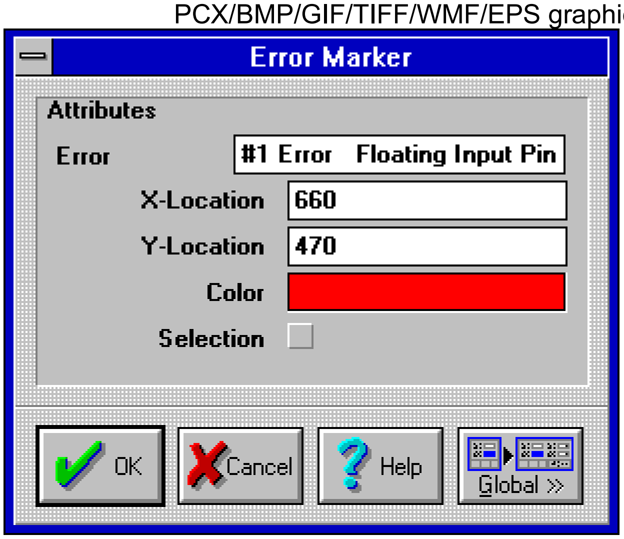
Place Ellipses Place elliptical shapes on the sheet



Place Pie Charts Place pie shapes on the sheet



Place Graphic Place PCX/BMP/GIF/TIFF/WMF/EPS graphical images on the sheet



Paste Array Define an array placement of clipboard contents

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 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

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 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

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 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

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 DECOMP.EXE must be in the PATH to Load Orcad Binary Library

The DOS application DECOMP.EXE is supplied with the OrCAD SDT 16 Bit system. Make sure that it is in your PATH.

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 more Space For Access Codes

Only 40 access codes can be added at one time

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 Editor Help

This on-line guide provides an introduction to the Advanced Schematic design system, including many of the features, key concepts and terminology used throughout the system. 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.

See also

Trace Netlist Report

Hierarchical and multi-sheet support

Advanced Schematic supports single sheet, multiple sheet and fully hierarchical designs. An entire multi-sheet project can be opened (or saved) with a single command. Projects can be navigated visually using the Project Manager tree display. A single mouse click moves the designer from sheet-to-sheet, even in complex hierarchies. Any number of multi-sheet projects or individual sheets can be open at any time.

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.

Special strings

Special purpose pre-defined strings allow the user to place date, sheet name, filename, component count or other information to be automatically generated when printing or plotting the sheet file. For example, a string called ".DATE_PRINT," when plotted, places the current system date on the plot.

Place these stringsbin the sheet, using the Place Text (not Text Frame) command. Special strings print using the font type size and color assigned during placement. Special strings include:

.DATE	Prints the date from the system calendar;
.TIME	Prints the time from the system clock,
.DOC_FILE_NAME	Prints the name of the schematic sheet file;

In the Options Sheet dialog box, the user can specify information which is added to the sheet title block when the ANSI title block option is selected. The Document button allows the user to include the organization name, address, sheet number, total sheets, title, number and revision. These will be ignored is the normal title block is selected.

Document options

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

Design verification tools

Electrical Rule Check allows quick verification of large or complex drawings. ERC checks are performed in accordance with user-specified physical and logical properties.

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 computer model of a circuit

Outwardly, the use of schematic capture is similar to the traditional drafting process, where graphical symbols for wires and components are rendered in drawings, which become a record of the design. Performing the process electronically provides many benefits in the form of automating the process of drafting, and more importantly, revising the design over time.

However, it's the capture part of the process that provides the main benefit -- the integral link between the design and production process.

Schematic capture also incorporates many data management facilities that exploit the capabilities of computerized design. For example, each Advanced Schematic sheet is an independent file. An automated system links these sheets when they are part of a common design project.

The Advanced Schematic component model
Libraries of component models
Connectivity
Wiring
Net Identifiers

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.

Netlists and connectivity

Wiring

The process of placing electrical objects in the sheet is often referred to as wiring. This is because the connectivity features allow you to work with electrical objects as though you were physically hooking-up the circuit.

Basic electrical objects, used in wiring your schematic include special wires, which are connective lines that carry signals or power between components; buses which graphically represent grouped nets; bus entries that graphically attach wires to buses; junctions which are used to connect crossing wires and parts that represent the component devices and their pins.

Two other special classes of electrical objects are provided as well: directives, which are used to indicate simulation points, unconnected pins (No ERC) and PCB layout attributes on individual nets and net identifiers, which are used to make electrical connections between sheets.

Net identifiers

Net identifier objects support connections between schematic sheets. These objects include:

net labels	Identify common nets on a sheet (or globally, across multiple sheets if the user specifies);
ports	Identify net connections between two sheets;
sheet entries	Identify net connections into a sub-sheet (referenced by a sheet symbol) and
power ports	Special symbols which represent a global power (or other user-specified) net.
hidden pins	Represent connections to global power (or other) nets, from placed parts.

Place Net Labels process
Place Ports process
Place Power Objects process
Place Sheet Nets process
Place Sheet Symbols process

See also

Hidden Pins
About netlists

Using connectivity

Connectivity is derived from the placement of certain electrical (or wiring) objects in the sheet workspace. However, not all wiring objects have connective behavior. Some objects use their physical geometry to establish connections. Other objects include logical connectivity in their behavior.

Physical connectivity
Logical connectivity

Physical connectivity

Physical connectivity is derived by placing two connective objects in physical contact. For example, a wire touching a component pin is deemed to be connected to that pin, and Advanced Schematic can extract that logical connection from the physical contact between the two items.

In general terms, when the "hot" points of any two connective objects "touch" they are deemed to be connected. However, there are some special rules that apply to certain classes of connections.

The Info Selected Pins command (shortcut: I, E) can be used to verify connections during schematic layout.

Wire to wire Wires that butt or have co-linear terminations are deemed to be connected. Co-linear wires that terminate elsewhere are not deemed to be connected. Wires that cross perpendicularly are not deemed to be connected unless a junction is placed at their intersection.

Wire to bus Buses are graphical representations of grouped signals only, and do not have any special connective properties. Net labels must be used to indicate connectivity on either side of the bus connection.Pins that touch or cross wires at any angle are deemed to be connected.

Wire to port Port connections are defined by their names (which reference a specific net name).

Wire to Sheet entry No special connectivity is present when wires and sheet entries are in contact. Connectivity is provided by the net label associated with the wire and the sheet entry net name.

Bus to object Buses are graphical representations of grouped nets only and have no special connective properties. Logical connectivity is used in these cases.

Net label to wire Net labels associate a wire with single net. To achieve this association, the net lable must be placed on the same grid point as the wire, either vertically or horizontally. Labels cannot be placed on 45 degree or non-orthogonal lines.

Net label to bus Buses are graphical entities and cannot be connectively net labeled.

Net label to pin Net labels cannot be directly connected to pins. The net label must be placed on a wire, connected to the pin.

Pin to pin Pins are deemed to be connected if they are in contact at any angle.

No ERC No ERC objects are deemed to be connected to pins or wires if they are in contact.

Logical connectivity

Logical connectivity depends upon the presence of net identifiers (net labels, ports, sheet entries, power ports and hidden pins) on the sheet. Logical connectivity does not require special placement or physical contact but relies on the matching of the net names that associate these objects within a single sheet or across multiple sheets in a project.

Design verification

Design verification is a general term for validating the electrical and logical contents of your design. A number of tools are provided that allow you to perform design verification from within the Schematic Editor, by generating various reports and by running the Electrical Rules Check (ERC) feature.

Because Window's multi-document environment allows you to switch between multiple application, you can generate netlists, reports and Electrical Rules Checks and examine these items without exiting the Schematic Editor or Library Editor.

Checking sheets and projects
Generating reports

Checking sheets and projects

While creating a schematic, a number of useful design verification features are available directly in the sheet workspace. The Edit Select Net and Edit Select Connection commands can be used to highlight all objects associated with a net or a single connection (component pins do not select in these cases). These commands are useful in verifying connections. The Info Components and Info Selected Pins commands display a list of all sheet components or all pins who are connected to the current selection.

Generating reports

Advanced Schematic design verification tools include four report generators which produce ASCII format text files:

Bill of Materials
Project Hierarchy
Cross Reference
Electrical Rules Check

Bill of Materials

The File Reports Bill of Materials (BOM) feature generates a report for all sheets in the current active project. Two versions of the BOM report are available. The (default) condensed pre-formatted version lists part type fields, description fields, quantities of each type and the designator fields associated with each type. This condensed version is produced in a tabulated ASCII format. The Expanded BOM format is produced in CSV (Comma Separated Value) format and includes several additional fields: 8 (internal) library component text fields, 8 part text fields and 4 package description fields. This format is intended for export to database or spreadsheet applications. You may need to specify CSV format when importing this BOM format to your database or spreadsheet.

About this process
About the CSV Editor

Project Hierarchy

The File Reports Project Hierarchy command (shortcut: F, R, P) generates a listing of project files for the current active sheet, hierarchical or multi-sheet flat project. This report is output in ASCII text format.

About this process

Cross Reference

The File Reports Cross Reference command (shortcut: F, R, C) generates a listing of part types and designator labels, and the sheet location (filename) for each item. The report is generated for the current active sheet, hierarchical or multi-sheet flat project. This report is output in ASCII text format.

About this process

Electrical Rules Check

Electrical Rules Check (ERC) report is a listing of electrical (and certain logical) violations and warnings for the current active project. A wide variety of basic electrical errors are reported. For example, open input pins on parts and shorts between two differently named power nets. The user can select the specific rules used to validate the project. No ERC symbols can be placed on sheets wherever the user prefers specific violations to be ignored by the ERC system.

An error or warning can be specified using a matrix of pin, port or sheet entry conditions.

Special symbols are overlaid on the sheet, indicating the location of the reported conditions. These symbols can be deleted individually as the design is checked.

The process of running an ERC is integral to producing a valid netlist for a project. The presence of electrical or logical violations will not prevent Advanced Schematic from generating a netlist, however incomplete or invalid. Carefully check and resolve all reported errors prior to netlist generation.

Setup Electrical Rules Check

The following options are provided in the Setup Electrical Rules Check dialog box:

Report Options

Multiple Net Names on Net - reports nets which have more than one net label associated with the physical net.

Duplicate Sheet Numbers - reports sheets with the same sheet number.

Duplicate Component Designators - reports components with the same reference designator (label) such as U1, etc.

Bus Label Format Errors - To be identified properly as a Bus label, the label must be in the following format: AB[0..9]. Errors that are reported include: Missing prefix; Missing bracket; Missing two dots (or decimal points); Numbers outside of the legal range (0-32000).

Supress Warnings - this option supresses reporting of Warnings. This allows you to generate a quick report of Errors only.

Output Options

Create Report File - Generates a text file which will be opened using the default text file application assigned in the File Run Setup dialog.

Add Error Markers - Generates graphical Error Markers which are placed in project sheets.

Net Identifier Scope

Specifies the use of net identifiers when creating a netlist (preparatory to generating an error report).

Pin/Sheet Entry/Port Rule Matrix

Sets the specific connection rules that are and the Error/Warning status for electrically connected items. Click inside the matrix to toggle through No Report, Error or Warning. For example, two output pins cannot be connected, so the two points on the matrix where these two items intersect are set to Error. Warnings can be supressed when generating ERC reports.

Report ERC process
Jump to Next Error process

Linking to other applications

Advanced Schematic provides a number of special tools for linking the product of schematic designs to other applications, from PCB design and circuit simulation -- to spreadsheets and databases. Topics include:

Netlists
Special links to PCB systems
PCB Layout directive
Back annotation
Cross probing by part
Cross probing by net
Cross probing by pin

PCB layout links

Advanced Schematic supports a number of links to PCB systems beyond basic netlisting.

PCB Layout directive The PCB Layout directive, object (supported by OrCAD SDT and PCB) allows the engineer to specify routing characteristics on a net-by-net basis. User definable fields for the PCB Layout object include routing track width and via size, layer and route priority. Protel for Windows Advanced PCB version 2 will provide comprehensive support for PCB Layout directives from Advanced Schematic.

Back Annotation The File Back Annotate command updates schematic part designators (U1:1, R32, etc.) based on a "was-is" list generated by the PCB layout package. Protel for Windows Advanced PCB 2 and a number of other layout packages support back annotation.

Forward Annotion Forward annotation is the process of implementing changes to an existing PCB layout from a schematic editor. This system will allow users to delete nets, add nets, delete nodes and add nodes at the schematic. Advanced PCB 2 will compare the current layout with the new netlist and remove existing routes on the PCB and convert these back into unrouted connections.

Cross-probing Protel for Windows fully supports multi-document and multi-application use of the Windows environment. This allows the user to open Advanced Schematic and a Protel for Windows PCB application at the same time. When a schematic sheet (or project) and its PCB layout are open at the same time, Advanced Schematic and Advanced PCB version 2 will support cross probing. For example, the user will be able to select a component in the schematic, and the PCB editor will jump to the same component, in real time.

Place Layout Directive process

See also

About netlists

Project management

A schematic design can consist of a single sheet drawing or multiple linked sheets. Irrespective of the number of sheets, Advanced Schematic treats each design as a project.

Multiple sheet projects support large or complex designs that cannot be served by a single sheet, irrespective of the sheet size. Even when the design is not particularly complex, there can be advantages in organizing the project across multiple sheets. For example, the design may include various modular elements. Maintaining these modules as individual files can be very convenient.

When two or more sheet files are associated or linked in some way, we refer to this as a multi-sheet project. There are a number of methods for organizing multiple sheet projects. Choosing one approach or another is based upon the type of design, its size and structure.

Project management is the process of defining and maintaining the links between the sheet files that make up a project. These links allow the user to open and save projects in a single operation. They provide a means of navigating, or viewing and accessing each individual sheet in a project and also support multi-sheet netlist generation and Electrical Rules Checks (ERC).

In Advanced Schematic, each sheet is stored as an individual file that is opened in its own independent window. Any sheet can be opened and edited independently of all other sheets, using the File Open Sheet command. Multi-sheet projects are opened using the File Open Project command.

Many Advanced Schematic features such as netlist generation, re-annotation and printing apply to the open sheet files associated with a project. Therefore, in many cases an entire project (which includes a Master sheet and all associated sheets) will need to be opened at once to perform these operations.

Four models of hierarchy
Using the Project Manager

Master sheet

All Advanced Schematic multi-sheet projects have a special sheet file called the master sheet. The master sheet is the top, or first sheet in the design hierarchy. The term hierarchy refers to the relationship between master sheets and other sheets (or **subsheets**) that make up the project. This hierarchical structure can have a number of forms, defined by the method used to connect the sheets together.

Hierarchical organization supports a truly modular approach by allowing the designer to work with functional blocks. These blocks have a spatial relationship on the sheets which natural support for either a "top down" or "bottom up" design methodology.

Subsheets

In multi-sheet designs, the master sheet always includes at least one sheet symbol. The sheet symbol is a special object that provides a graphical representation of another sheet (called a subsheet) in the hierarchy.

Along with its graphical display attributes (color, size, location, etc.) the sheet symbol has two additional fields: a sheet name and a file name. The sheet name field is a text label, and is provided for reference only. The file name refers to a specific sheet file that the sheet symbol represents and provides a link between the **master sheet** and its subsheet, and thereby defines a project.

Four models of hierarchy

Hierarchies can have two levels or many levels, depending upon the hierarchical model and the method of intersheet (electrical) connection. This is accomplished by defining the scope of **net identifiers**. Advanced Schematic supports four models of intersheet connections, defined by the scope of three special net identifier objects, which provide the connective "glue" that defines the sheet-to-sheet hierarchy.

The relationship between project sheets and net identifiers is best described by illustrating the four possible models of project organization under Advanced Schematic.

1. Global ports define intersheet connections
2. Global net labels and ports
3. Sheet entries define simple hierarchies
4. Complex hierarchies defined by sheet entries

See also

More about hierarchical design

Global ports define intersheet connections

This model for intersheet connections uses global ports. This model is sometimes referred to as a "flat" design. Global ports point to a net on another sheet. The **Master sheet** includes a sheet symbol for all sheets in the project, however no connections are made into these symbols on the master sheet. Net names in each sheet are local, meaning that the net name applies across a single sheet only, not other project sheets. In flat sheet designs, each sheet is always unique.

Ports route connections that pass from sheet to sheet. This model of sheet organization treats your design as though it were laid out on a single large sheet that has been cut into individual pages. While this approach works fine for designs of limited size, management of large designs can be somewhat awkward, as this method, relies on the designer to maintain unique labels for each pair of ports that define a single intersheet connection.

Using net labels and ports to implement inter-sheet connectivity

See also

More about hierarchical design
OrCAD net identifiers

Global net labels and ports

In this hierarchical model, intersheet connections are provided by global net labels. This model supports multi-sheet designs from "flat" schematic capture systems, such as Protel Schematic 3 (DOS). In Advanced Schematic, global ports can also be used with this model, providing the user takes care to avoid conflict between port and label names. A sheet symbol for each sheet must be placed in the **Master sheet** to allow the Project Manager to associate (or link) the **Subsheets** as a single project.

Using net labels and ports to implement inter-sheet connectivity

See also

More about hierarchical design
OrCAD net identifiers

Using net labels and ports to implement inter-sheet connectivity

The sheet symbols in flat projects are not connected to nets. Flat design intersheet connections are accounted for on each sheet separately, using either net label, ports or both net labels and ports together.

Placing a sheet symbol does not create a new sheet, it only establishes a link to a sheet with the same filename.

Place Ports process
Place Net Labels process

Sheet entries define simple hierarchies

This hierarchy model is sometimes referred to as simple hierarchy. Simple hierarchy supports multi-level or block design, where the hierarchy can be represented by a tree-like structure and all intersheet connections are vertical, defined by sheet entry symbols in each sheet symbol. In this model, the sheet symbol represents a child sheet, which descends from the parent. In simple hierarchies, each sheet symbol is used once in the design. In other words, each sheet in a simple hierarchy is unique. Net identifiers are local, not global, in this model.

Using sheet symbols to implement hierarchy
Master sheet
PlaceSheetSymbolsPlace Sheet Symbol process
AddSheetNetsPlace Sheet Entry process

See also

More about hierarchical design
OrCAD net identifiers

Complex hierarchies defined by sheet entries

This design model is sometimes referred to as complex hierarchy. The same sheet symbol can be placed more than once in a project, either by being placed more than once on a single sheet, or by being placed on multiple sheets.

This model fits projects that are highly modular. An example would be a stereo amplifier, where left and right channels are identical circuits.

Hierarchy in Advanced Schematic is not exclusively linked to the electrical links that make up a multi-sheet circuit. You can also create projects which group unlinked sheet files for convenience.

Using sheet symbols to implement hierarchy
Master sheet
PlaceSheetSymbolsPlace Sheet Symbol process
AddSheetNetsPlace Sheet Entry process

See also

More about hierarchical design
OrCAD net identifiers

Using sheet symbols to implement hierarchy

The sheet entry symbol, links the subsheet to a net on the parent sheet. Every subsheet has a "parent" sheet, either the master sheet, or another subsheet from which it descends.

Similarly, so-called "flat" design master sheets (the first two models, above) include sheet symbols, labeled with the filenames of each subsheet. These sheet symbols are not "wired" together, using sheet entries.

Place Sheet Symbol process
Place Sheet Entry process
Create Sheet Symbol from Sheet process
Create Sheet from Sheet Symbol process

More about hierarchical design

So-called "simple" ane "complex" hierarchical models show the most powerful way to organize multiple sheet designs. In both of these models, sheet symbols represent functional blocks, with the sheet entry symbols serving as connectors that tie circuitry on the sheet to the sub-sheet. This hierarchical structure can be represented by thinking of the first sheet as the "parent" and the sheet represented by sheet symbols as the "child." In the terminology of hierarchical design, we can say that the child is descended from the parent. Naturally, the child can have its own "children," -- additional sheets that descend, in this top-down structure, to lower and lower levels.

Hierarchy can be either simple, where each sheet is unique or complex, where the same child sheet (and its children) can appear more than once in the design -- a modular approach.

Because each Advanced Schematic sheet is opened in its own window, you can display any number of sheets simultaneously (memory permitting). Moving from sheet to sheet can be a simple as clicking in the window to make it active.

This association of sheets resides on two levels. At the project level, the association is maintained by the presence of sheet symbols in the master sheet. Projects are also defined at the electrical (or connective) level by net identifiers. Net identifiers provide the links that connect circuits across multiple sheets.

OrCAD project management

Using the Project Manager

Advanced Schematic supports single sheet, multiple sheet and fully hierarchical designs. An entire multi-sheet project can be opened (or saved) with a single command. Projects can be navigated visually using the Project Manager tree display. A single mouse click moves the designer from sheet-to-sheet, even in complex hierarchies. Any number of multi-sheet projects or individual sheets can be open at any time.

Opening schematic files
Opening hierarchical or multi-sheet projects
Adding and removing individual sheets from a project
Simple or complex hierarchical designs
Save Project and Save All commands
Navigating projects
Loading OrCAD files
Loading Protel (DOS) schematics
Create Sheet Symbol from Sheet process
Create Sheet from Sheet Symbol process

Opening schematic files

Under Advanced Schematic you can load any number of schematic sheet files into separate windows using the Multiple Document Interface (MDI). Advanced Schematic will load the following formats: Advanced Schematic (text and binary); Protel Schematic (DOS version 3) and OrCAD SDT (version 3 or 4) 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 files with the (default) Advanced Schematic file extension of .SCH. 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
Open Project process

Opening hierarchical or multi-sheet projects

The File Open Project command is used to open all sheet files associated with a hierarchical or multi-sheet project. To open a previously created project:

> Type the filename for any project sheet (include the full path, if different from the path currently listed). If the project is hierarchical, you can use any file in the project to open that sheet and any sheet that is lower in the hierarchy;

You can also double-click on the master sheet filename in the Files window. To change directories, double-click on any the available options in the Directories window.

This command can be used at any time and any number of hierarchical, flat and single sheet projects can be open simultaneously, limited only by available memory.

As a project loads, an icon for each sheet file is placed in the Project Manager window as a sheet window opens for each file inside the Schematic Editor window. These sheet file windows will cascade to reveal the sheet name in each title bar.

Any number of projects and individual sheets can be opened and displayed in the project window, limited only by available memory. If the window fills, a scroll bar is added to the project window.

File Open process
Open Project process

Adding and removing individual sheets from a project

The contents of a multi-sheet project are defined by sheet symbols placed in the topmost or master sheet. To add an existing sheet to a project, place a sheet symbol in the master sheet with a matching sheet filename field. To remove a sheet from the project, simply delete the sheet symbol with that sheet's filename.

File New process
Create Sheet Symbol from Sheet process
Create Sheet from Sheet Symbol process

Loading files (OrCAD)

Flat designs

OrCAD "flat" projects automatically create their own master sheet the first time they are loaded as a project (Open Project command). Sheet symbols will be placed in the master sheet for each subsheet, based upon the "pipelink" strings placed in the original OrCAD root schematic.

Simple or complex hierarchical designs

Simple and Complex "hierarchical" projects load just like Advanced Schematic files. Master (or root) and subsheet references are maintained by the OrCAD sheet symbol filenames, sheet net symbol names and net labels.

In Advanced Schematic, all OrCAD design object types can be displayed on the sheet at all times, including simulation and layout directive objects.

Warning: Saving files in OrCAD (SDT 4) binary format will cause the loss of primitives and other file information that is not supported by the OrCAD format. Lost information will include vector graphical objects placed in the sheet, design object text fields exceeding 128 characters, imported images, color and font assignments, etc. Consult your OrCAD documentation to determine supported data types and limits.

Loading files (Schematic 3)

Protel Schematic 3 (DOS) files are handled similarly to "flat" OrCAD projects. If the files have a common root name, e.g., CONTROL, then all files that are named CONTROL.S01-.S99 will be included in the new project. A master sheet, with sheet symbols referencing each file will be automatically created.

For example, if you use OPEN Project on DEMO.S01 the DEMO.PRJ will be created and DEMO.S01, DEMO.S02 will be loaded. A Sheet symbol for each will be placed in DEMO.PRJ.

File Open process

Save Project and Save All commands

Save Project Edit Save Project compliments the Open Project command, updating all the open sheet files, associated with the current active project, in a single operation. To use this command, first move the focus to any sheet in the project by clicking directly on the file's open sheet, or the icon representing the sheet in the Project Manager window.

To save all the sheet files that constitute a single hierarchical or multi-sheet flat project, use the File Save Project command. Sheets will be saved using the default Protel binary format. All sheets that belong to the project will be updated.

Save All The Edit Save All command extends this concept, updating all open files, whether or not they are part of the current active project.

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

The Save All command (shortcut: F, L) can be used to save all of the printed circuit boards in all currently opened windows. All files will be saved in binary format.

File Save process
File Save All process
File Save As process
File Save Project process

Navigating projects

When you use the Open Project command, click the Project Manager button on the main tool bar or choose the Options Project Manager command, the main Advanced Schematic application window displays a special project window to the left of the workspace. This window is used do display all the current schematic master sheet and sheet files that are currently opened. The active sheet file icon is highlighted.

You can click on any sheet icon to move the current focus to that sheet window. If the file is currently opened, the file window will be displayed in the Schematic Editor window. If the file has been closed or minimized, the window will be opened. Double-clicking on the file icon will maximize the sheet file window.

Toggle Project Manager

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.

Change process

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.

Place Parts process
Run Library Editor process
Edit Component from Browser process

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.

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.

Run Library Editor process
Edit Component from Browser process

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.

Run Library Editor process
Edit Component from Browser process

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.

Run Library Editor process
Edit Component from Browser process

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.

Update Parts from Library process

Cross Probe Part (PCB) process
Increment Part Number process

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.

Run Library Editor process
Edit Component from Browser process

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.

Run Library Editor process
Edit Component from Browser process

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.

Run Library Editor process
Edit Component from Browser process

Default designator

A default designator prefix can be stored with each library item. When components are placed, designators will automatically include this prefix.

Run Library Editor process
Edit Component from Browser process

Subparts

Multi-device components include graphical descriptions for each subpart. Each can be independently modified in the Schematic Library Editor.

Run Library Editor process
Edit Component from Browser process
Increment Part Number process

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.

Run Library Editor process
Edit Component from Browser 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.

Run Library Editor process
Edit Component from Browser 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.

Run Library Editor process
Edit Component from Browser 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.

Run Library Editor process
Edit Component from Browser process

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.

Run Library Editor process
Edit Component from Browser process

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.

Run Library Editor process
Edit Component from Browser process

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.

Run Library Editor process
Edit Component from Browser process

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.

Run Library Editor process
Edit Component from Browser process

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.

Run Library Editor process
Edit Component from Browser process

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.

Library Add/Remove process

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

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.

Make Project Library process

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.

Updating Parts from Library process

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

New Opens 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;

Open Opens an existing library file. The normal mask and path specifications can be used to screen for specific library types, by extension;

Merge Opens a second library and adds all components to the current active library file;

Close Closes the current library; If the library has not been saved (or saved as) the prompt "Save Library before closing?" will be displayed;

Save Saves the current active library;

Save As Saves a renamed version of the current active library;

Save All Saves all open library files;

Plot/print Prints the content of the current active Library Editor edit workspace;

Report Generates an ASCII text file report of all component type descriptions in current active library;

Exit Quits 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. Advanced Schematic 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-Sheet. Other buttons are provided for toggling the show/hide status of other toolbars and panels.

Drawing tools Drawing tool buttons provide access to the Place commands for graphical objects: lines, text, ellipeses and arcs, rectangles, etc.

Wiring tools Buttons on the Wiring toolbar provide access to the Place commands for electrical objects: wires, buses, bus entries, components, junctions, etc.

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 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
Sheet options
Workspace options

About Preferences

Advanced Schematic stores many user settings, such as printer/plotter setups, display colors, and many other options in a special file called SCH.INI that is automatically added to the Windows directory, the first time that you Exit the application. Thereafter, SCH.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 Advanced Schematic SCH.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
Other preferences

Sheet options

The Options Sheet dialog box is used to assign colors to the sheet and sheet border/title block. These assignments are saved with the sheet and are also used as defaults when a new file is created.

Workspace options

The Options Preferences command is used to set-up the workspace design environment.

Options dialog box
Workspace colors
Saving default settings
Auto Backup
Grid options
Other preferences

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 SCH.INI and LIBEDIT.INI files.

Sheet Options process

Other preferences

Along with color assignments for the workspace, the Options Preferences command (shortcut: o, p) sets-up a number of environmental conditions for Advanced Schematic. Options include:

Cursor Shape

Visible Grid

Auto Pan

Question Delete

Snap to Center

Save Defaults

OrCAD Port Length

Drag Wires

Auto Backup

See also

Preferences
About Preferences

Schematic Editor workspace

This section describes the workspace for schematic/library design.

Sheets

When you choose the File New command, an empty sheet is displayed. This is where your design is displayed. Sheet windows are titled with the current path and file name, if any.

Borders

Protel provides two default borders. Use either the standard border or the ANSI standard border that follows the ANSI reference grid convention (for additional information, please refer to ANSI standard Y14.1-1980). You also have the option to turn-off the supplied border and use Advanced Schematic's drawing tools to create a custom border. You should be aware that not all devices can print all the way to the edge of the page. This can make it impossible to include all of the standard or ANSI borders when printing at 100% scale. You can change the print scale to accommodate the maximum printable area of your printer.

Title block

Protel provides two default title block formats. You can use the either standard title block or the ANSI standard title block that is somewhat larger. Some of the information in the title block is provided automatically, e.g., the sheet size, file name and creation date. Other information can be added using the Place Annotation and Place Text Frame commands. You also have the option of hiding the supplied title block and use Advanced Schematic's drawing tools to create a custom title block.

Library Editor workspace
Coordinate system
Sheet navigation
Changing your view of the sheet
Cursor control
Grids
Sheet Options process

Choosing a sheet size

Choose the Options Sheet command to open the Sheet Options dialog box to choose from one of the available sheet sizes or define a custom sheet.

Advanced Schematic allows you to choose from 10 standard imperial and metric sheet sizes, or to define a custom sheet size up to 65 inches (1650 mm) square.

Schematic sheets are conventionally displayed and printed in "landscape" (wide) rather than "portrait" (tall) orientation. Advanced Schematic allows you to display and print your drawings in either orientation.

The maximum available work area in a sheet (with the border hidden) will depend upon the target device. Many printers and plotters cannot print to the edge of the sheet, so some trial and error may be necessary to determine the exact available work area. Because of this, the standard ANSI and ISO border specifications cannot be applied when targeting these devices. Advanced Schematic can compensate for this by allowing you to scale the output during printing or plotting.

A smaller work area will be available when sheet borders are displayed. The default sheet boarder removes .2 -- .4 inch (approximately 5 - 10 mm) from the working area, depending upon the sheet size selected.

Sheet Options process

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.

See also

Schematic Editor workspace

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

Sheet navigation

This section describes how processes and process launchers allow easy navigation of the sheet.

Search For
Jump
Using Location markers

Search For

The Edit Search For command allows you to conveniently locate a specific component, net or text string without having to zoom, pan or scroll through multiple screens. Search For commands include:

Component Type the designator in the Search For dialog box. If you do not know the designator, type ? and press ENTER or click LEFT MOUSE to scan the board for all placed components; Choose from the Components Placed dialog box.

The cursor will jump to the center of the reference pin (usually pin 1 or A1) of the selected component.

Net Type the net name in the Search For dialog box. If you do not know the net, type ? and press ENTER or click LEFT MOUSE to scan the board for all nets; Choose from the Nets Loaded dialog box.

The cursor will jump to the nearest pin that belongs to the selected net.

Pin Choose a placed component to open the Jump to Pin Number dialog box; Type the pin number. The cursor will jump to the center of the named pin.

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.

Search for Net
Search for text
Search for And Replace text
Search for next instance of text

Jump

The Jump command options allow you to quickly move the cursor to a pre-determined coordinate. 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 layouts.

Origin

Jumps 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).

New Location This option allows you to type in the desired coordinates for the jump.

Type an X coordinate (a distance from the left hand side of the work space), the default is the current X position. Type the desired location and press ENTER or LEFT MOUSE.

Type a Y coordinate (a distance from the bottom of the work space), the default is the current Y position. Type the desired location and press ENTER or LEFT MOUSE. The cursor will now be moved to the specified location.

Jump Location process
Jump Origin process
Jump to Error Marker process

Using Location markers

Ten user-definable location markers are provided, which allow the user to move to a pre-determined position in each sheet, with a single command. These markers can be placed in the sheet at any time, using the Place Location Mark command. The coordinates are stored for each of ten locations.

When any of the location markers have been set by the user (see above), the Edit Jump command can be used to re-draw the screen at the center of the pre-set location. The current zoom setting remains unchanged when jumping, so it may be desirable to zoom-in prior to jumping to the new location.

Set Location Mark process
Jump Location Mark process

Jumping to locations

Changing your view of the sheet

As you click to move from document window (sheet) 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. Pressing ctrl+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.

File

File menu commands are used to create new files, open files or entire projects, close files, save the current file, save the current file under as new filename, save all open files, print or plot the current file, export a file in a new format, generate reports for the current files or exit the Advanced Schematic program.

Edit

Edit menu commands are used to make changes to the current sheet window, including: Undo or redo actions; cut, copy, paste or clear a selection; make selections of items in the window; de-select items; delete items; change items; move items; search for items; jump to sheet locations.

Place

Place menu commands are used to place graphical and electrical items in the current sheet document window.

Library

Library menu commands are used to browse libraries and open the Library Manager.

A comprehensive library of manufacturers components is included in the Advanced Schematic package. The Advanced Schematic library system also includes a complete set of commands for creating, editing and using library components. The user can create different component libraries for various uses. Multiple libraries can be opened simultaneously.

Options

Options menu commands are used to change system defaults and user preferences, display attributes for objects; to assign hot keys to menu commands; and to toggle display elements, e.g., scroll bars, on or off.

Zoom

Zoom menu commands are used to change the display area in the current document window.

Info

Info menu commands are used to check the current system status; current design statistics; list the components in the current sheet and list the components and pins associated with a selected net.

Window

Window menu commands are used to re-arrange or close all of the current open document windows.

Help

Help menu commands are used to open any Windows help file, direct the user to specific help topics and provide information about the current application.

Details about individual menu commands can be found in the Advanced Schematic Reference and the On-line Help system.

Toolbars

Many Advanced Schematic menu commands 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
Wiring tools
Drawing tools
Library tools
IEEE tools

Main tool bars

The main toolbar, displayed across the top of the Schematic Editor and Library Editor application windows, includes a number of general purpose tools. Differences in the two applications reflect the special requirements of sheet level and library level editing. These fixed tool bars cannot be repositioned by the user, however they can be displayed or hidden from the Options menu.

Toggle Main Toolbar

Wiring tools

Includes tools for creating and placing objects that represent the electrical elements of a schematic. Electrical objects are objects that work with connectivity features.

These include wires, buses, junctions, parts. Advanced schematic recognizes the connections that exist when these items are graphically joined, and uses this information to generate netlists, maintain connections when items are moved and perform electrical rules checking.

Toggle Wiring Toolbar

Drawing tools

Includes tools for creating graphical objects, including single line text and text frames. Graphical objects include items that are not regarded by these connectivity features.

Examples of non-electrical items include lines, polygons, bezier curves, rectangles, text frames, etc. Graphical objects are used a general purpose drawing and documentation tools for your schematic.

Toggle Drawing 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.

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.

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
Special 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 wires
Polyline behavior
Creating other objects

Object tutorial - Placing wires

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 wire in the current sheet window:

Zoom in on the sheet (press PGUP two or three times) until you can see the visible grid. Choose the Wire command from the Place menu (shortcut: W, P or click the Wire tool button);

The prompt "Select Wire 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 wire;

The prompt "Place Wire" is displayed on the status line.

Drag the wire 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 wire and that the prompt "Place Wire" is still displayed.

Move the cursor to continue with a new wire 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 wire 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 wire;

Note that "Place Wire" is still displayed on the Status line. This allows you to end one series of connected wires and then begin a new series of wire segments elsewhere in the workspace without having to choose the Place Wire command again.

Place Wire process

Polyline behavior

The segmented line created when you place a graphical line, wire or bus 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

Wire/bus placement mode
Selection and Focus

Line placement mode

Advanced Schematic provides two line, wire and bus 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.

Error markers

Error markers are placed in schematic project sheets automatically during ERC (Electrical Rules Check) generation. These objects cannot be placed manually. A 360 degree arc is placed at the site of each connective error or warning. The ERC error markers can be deleted from the sheet by clicking over the object with LEFT MOUSE and pressing DELETE.

Report ERC process
Jump to Next Error process

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) or set focus process
Change (Graphically) or move 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
Making selections
Selection and Focus
Select and De-select commands
Toggle Selection
Using the clipboard - Cut/Copy/Paste
Moving items
Selection and other Windows commands
Importing selections

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
Net
Connection
Toggle Selection

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.

Special criteria are used to search and replace strings in all primitives that have text fields.

Aligning items
Object colors
Using text field Wildcards

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
Place Text Frame 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

Completing the schematic layout is only part of the design process. In most cases you will need to generate permanent drawings that can be filed and viewed "off-line." Advanced Schematic includes support for a wide variety of "hard copy" options for this stage of the design process. Virtually any device that is supported by Windows 3.x can be used to print or plot your drawings.

Advanced Schematic provides a wide range of output options when you are ready to turn your layout into artwork.

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 problems

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.

Running Electrical Rules Checks

The File Reports Electrical Rules Check command runs the Electrical Rules Check (ERC) process and generates a listing of electrical and logical violations and warnings for the current active sheet, hierarchical or multi-sheet flat project. A wide variety of basic electrical errors are reported. For example, open input pins on parts and shorts between two differently named power nets. The user can select the specific rules used to validate the project. No ERC symbols can be placed on sheets wherever the user prefers specific violations to be ignored by the ERC system.

An error or warning can be specified using a matrix of pin, port or sheet entry conditions.

When the report is generated, special symbols are placed on the sheet, indicating the location of the reported conditions.

The process of running an ERC is integral to producing a valid netlist for a project. The presence of electrical or logical violations will not prevent Advanced Schematic from generating a netlist, however incomplete or invalid. Carefully check and resolve all reported errors prior to netlist generation.

This feature was still under development at the time this guide was printed. Please refer to the README.TXT file and On-line Help system for further information.

Netlists

Netlists provide the **"capture"** of the schematic capture process. Normally, netlists are text files that list the component parts and pin-to-pin connections that allow schematic information to be imported into a simulation or layout environment. Simply stated, a netlist is a summary of all the connections (or networks) that comprise a circuit.

Generally, netlists are simple ASCII text files. The typical netlist format includes descriptions of components, such as the designator and package type combined with the pin-to-pin connections that define each net. Loading a netlist into a printed circuit board layout package automates many of the tedious and error prone operations inherent in the design process. In Advanced Schematic, you can quickly generate and examine a netlist of your current design without leaving the editor.

Generating netlists
Connectivity and netlists
About netlists

See also
Trace Netlist Report

Connectivity and netlists

A key element of electronics design automation is the ability of schematic capture and PCB layout systems to recognize the connections within a circuit. This concept -- connectivity -- is used at several levels during the design process. Two types of connectivity are employed under Advanced Schematic:

Logical Logical connectivity is the connection information derived from netlisted information. For example, the pin-to-pin connections in your schematic are transferable, via a netlist, to the PCB editor. In Protel for Windows, these connections are stored and used during the layout process.

Physical Protel for Windows uses the physical geometry of your layout to perform connectivity-based operations. One example is the ability to select a connection or net. Physical connectivity is used, along with logical connectivity, when generating a netlist from a design. It also allows wire-to-pin connections to be maintained as components are moved, etc.

See also

Trace Netlist Report

About netlists

Netlists come in many different formats, but are usually generated as ASCII text files that carry two basic types of information:

1. Descriptions of the components in the circuit;
2. A list of all pin-to-pin connections in the circuit.

Some netlist formats combine both sets of data in a single description, Others, including Protel, separate the two data into separate netlist sections.

As straight-forward text files, netlists are readily translated into other formats using a simple, user-written program. Netlists can also be created (or modified) manually using a simple text editor or word processor.

If you intend to manually edit a netlist, make sure that you save the results in an "un-formatted" or "text only" form, as hidden "control characters" can render the netlist un-readable by Protel for Windows.

Some netlist formats can include additional information in component text or net text fields. This information is used to link netlists with simulators and board layout and routing in ways that aren't covered by basic connectivity.

See also

Trace Netlist Report

Protel netlist format

The standard Protel netlist format is a simple ASCII text file, split into two sections. The first part of a Protel netlist describes each component:

[	marks the start of each component description;
U8	the Component Designator (label);
DIP6	the Package Description (pattern). An identical description (or Type) will be required in the PCB library;
74LS38	is the Comment, (or value);
(blank)	three lines are left blank for future provision;
(blank)	
(blank)	
]	marks the end of the component description.

The component description section is followed by a listing for each net within the netlist:

(	marks the start of each net;
CLK	is the name of the net;
U8-3	shows the first component (by designator) and pin number. Pin numbers in Protel for Windows library components must be an exact match;
J2-1	indicates the second node in the net;
U5-5	indicates another node;
)	marks the end of the net.

Note that net descriptions are distinguished from component descriptions by the use of rounded, rather than square brackets. The extension .NET is reserved for Protel format netlists.

An extended version of this netlist, called **Protel 2**, supports additional information for simulation and PCB layout.

Netlist parameters

The following parameters apply to Protel netlists:

Protel (1) netlist format

Designators and Package Descriptions (Type) are limited to 12 alphanumeric characters. Comment fields can be up to 32 characters long. Net names can be 20 characters. Pin numbers in netlists are limited to 4 alphanumeric characters. Blank spaces (or other hidden characters) are not allowed in designators, package descriptions and pin numbers.

This format is supported by Protel Autotrax and Protel for Windows PCB layout systems and Tango PCB.

Protel 2 extended netlist format

All fields can be any length, up to 255 characters. Blank spaces (or other hidden characters) are not allowed in designators, package descriptions and pin numbers.

Any number of components or nodes can be included in a Protel for Windows netlist, limited only by available memory.

This format is supported by Protel for Windows Advanced PCB version 2.0 or later.

Other netlist formats

For information about field length limits and other constraints in other netlist formats, please refer to the appropriate manufacturer documentation.

About netlists

See also

Trace Netlist Report

Other netlist formats

Netlists generated or used by systems other than Protel) normally have many similarities to the Protel format. These are universally ASCII text formats that include component descriptions and net listings. However, the order in which component or net information is displayed may vary, and package names (e.g., DIP6), component designators and Pin identifiers may require editing to match Protel for Windows field restrictions. Often, translation of the netlist is an option in the schematic package. Netlists created using either a Protel or "Tango" output option will usually be fully compatible with Protel for Windows.

Package description (type or footprint) names and pin numbers must have exact matches in the Protel for Windows library for all components and connections in the netlist. Protel for Windows accepts either a dash (-) or comma (,) delimiter between the designator and pin number (e.g. U-6 or U,6).

Generating other netlist formats
About netlists
Netlist format notes
Generating SPICE netlists

Netlists and PCB layout

Protel for Windows PCB systems (and some other board layout packages) report missing component patterns or missing pins when two factors are present in the netlist:

One or more Package Descriptions (or Type) is missing from schematic component information in the netlist, or the package in the schematic does not match any Protel for Windows library component. The names of missing components and pins will be listed in a Netlist Report file, if desired;

It may be necessary to re-edit the schematic or netlist to include the Type information, or additional Protel for Windows library components may be created to match any unique descriptions in the netlist.

If all components are present but pins are reported missing, the cause is usually that the schematic package's pin numbering differs from Protel for Windows.

Schematic libraries contain specific components and devices. PCB footprint (decal) libraries contain generic footprints which can belong to various specific components -- each having different pin assignments.

For example, a transistor shape can represent various combinations of "E," "B" and "C," -- each of which must be assigned to the correct pin number in the PCB layout system. Capacitors are a similar case, with pins often named "A" and "K" in the schematic.

One solution is to leave the Schematic pin designations as "E," "B" and "C" and then place components on the PCB and change the pad designators to match. If you have a lot of pin outs in the same orientation, you may want to make a special version of the component in the library using the correct pin identifiers.

Generating netlists

You can generate a netlist for a project at any time while using the Schematic Editor File Create Netlist command. When you choose this command, Setup for Netlist dialog box opens.

Output format
Net identifier scope
Protel netlist format
Netlist parameters

About netlists
Other netlist formats
Netlist format notes
Generating SPICE netlists
Create Netlist command

See also
Trace Netlist Report

Output format

Advanced Schematic currently supports the following netlist output formats:

Protel (original format)

Protel 2

Protel Wirelist

EDIF 2.0

Algorex

Applicon Leap

Applicon Bravo

Cadnetix

Calay

CBDS

Computervision

EE Designer

Futurenet

HiLo

Intergraph

Mentor Boardstation 6

Multiwire

OrCAD PCB II

OrCAD PLD

PADS ASCII

PCAD

PCAD nlt

Racal Redac

Scicards

Spice

Tango

Telesis

Vectron

About netlists
Other netlist formats
Netlist format notes
Generating SPICE netlists

Net identifier scope

Three options are provided that define the method used to associate netlist identifiers (ports, net labels and sheet entries) with nets on each project sheet. The method used to identify global or local nets, defines the netlist contents. The three options are:

Net Labels and Ports Global

With this option, net labels are assumed to apply to all sheets in a project. In other words, nets are global and each instance of a net name or port is deemed to be connected to all other identically named identifiers. This model works like Protel Schematic 3 (DOS), where netlabels are always global to all project sheets.

Only Ports Global

This option treats net labels as local only. All identically named nets on a sheet are deemed to be connected. Intersheet connections occur only through identically labeled ports. This model works like the OrCAD SDT "flat" project model.

Sheet Symbol / Port Connections

This option makes intersheet net connections only through sheet symbol entries and subsheet ports. Ports are deemed to be connected only to identically named nets entering through their sheet symbols on parent sheets. This model works like the OrCAD SDT "hierarchical" project model.

About netlists

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.

OrCAD commands
STD Sheet files
SDT Design Objects
SDT libraries
SDT Utilities
Backward compatibility to SDT

OrCAD   sheet files

When you open an OrCAD SDT 3/4 file in Advanced Schematic, all SDT 3/4 design objects and data are available for editing. These sheets can be edited, saved in OrCAD SDT 4 format then re-opened and edited in SDT 4.

See also

Loading files (OrCAD)

Backwards compatibility to OrCAD SDT

Advanced Schematic supports all OrCAD SDT 3/4 design objects and other database elements when you open OrCAD sheets. When you save an Advanced Schematic file in OrCAD SDT format, most sheet elements are supported at some level. However, Advanced Schematic primitves can include attributes that are not supported by OrCAD, for example, color or font assignments for individual objects and text fields longer than 128 characters. This information will not be recognized by OrCAD, but you should still be able to save a compatible OrCAD format file in virtually every case. Fully-compatible means that the feature will be translated back to its normal OrCAD version, without the need to perform any manual editing or clean-up.

Saving OrCAD files

Organization (title block).

Junctions

Wires and buses

Dashed lines

Bus entries

Module Ports

Net Labels

Sheet Symbols and Sheet Nets

Parts

Power Objects

Trace Name

Vector Column

Stimulus

Error Markers

No Connect

Layout Directive

Libraries

Loading OrCAD schematics

Connectivity

Warning Saving files in OrCAD (SDT 4) binary format will cause the loss of some file and primitive information that is not fully-supported by the OrCAD format. Lost information may include some graphical objects, design object text fields exceeding 128 characters, imported images, color and font assignments, etc. Consult your OrCAD documentation to determine supported data types and limits.

OrCAD Design Objects

OrCAD Design Objects types are listed below, along with Advanced Schematic equivalents. Other OrCAD terms and procedures (and the Protel equivalents) are defined in the Glossary and relevant sections of the User Guide and Reference Manual.

OrCAD object	Protel object
Module Port	Port
Sheet Symbol	Sheet symbol
Sheet Net	Sheet entry
Power Object	Power port
No Connect	No ERC (electrical rules check)
Trace	Probe
Vector	Test Vector Index
Stimulus	Stimulus
Layout Directive	PCB layout
Tag	Location marker
Part	Part (e.g., device, what is placed)
Pipelink	Not required (place sheet symbol in master sheet).

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.

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 commands
Schematic 3 libraries
Block, highlighting, etc
Utilities

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.

PROTELLIB
Other bitmap components
Pin Editing
Connector components
Advanced Schematic Libraries

PROTEL.LIB (Schematic 3)

This library contains substitute vector parts for all bitmap components from the standard libraries delivered with Protel Schematic 3.3. When you load Schematic 3.3 sheet files, these components will be substituted automatically for the bitmap versions.

User created bitmaps (Schematic 3)

Protel Schematic 3 libraries and sheets may include user-generated bit map graphics that are not substituted when Schematic 3 sheet files are loaded into Advanced Schematic. Advanced Schematic creates and substitutes a block-type representation of this component parts when loading these files. These parts are fully functional, but will not include details from the user-created bitmaps.

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.

Block, highlighting, etc (Schematic 3)

In Advanced Schematic the functions of the Block and Highlight commands (such as Block Define or Highlight Net) have been merged into the concept of selection. If you wish to perform an operation on a group of objects on the sheet, you first select the items, then use one of the Edit menu commands.

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.   Complete descriptions of the following file formats are documented in the Advanced Schematic User Guide.

- Schematic Editor File Format
- Schematic Library Editor File Format
- SCH.INI Format
- LIBEDIT.INI Format

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.

OrCAD SDT Help

Again (OrCAD)

Advanced Schematic Editor does not have an equivalent to the Again command.

Block (OrCAD)

Move
Drag
Fixup
Get
Save
Import
Export
ASCII Import
Text Export

Move (OrCAD)

To move selected objects,

Choose Move and then Move Selection from the Edit menu.

Make sure that the current selection includes only those objects you wish to move. You can use the Edit Select and De-Select menu commands to select and de-select objects.

See also

MoveSelection

Drag (OrCAD)

To move objects and still maintain bus/wire connectivity,

Choose Move and then Move from the Edit menu.

Wires will retain their orthogonality as objects are moved around the worksheet. You can also use the shortcut CTRL+LEFT MOUSE to pick up and move an object.

See also

Move

Fixup (OrCAD)

The Change Object Graphically command can be used to move and clean up individual bus and wire segments. There is not a specific menu command for this process.

While moving wires or bus segments you can use the SPACEBAR to toggle through the different angle modes of wire and bus placement. The INSERT and DELETE keys can be used to add and remove line segments.

See also

Change Object Graphically

Get (OrCAD)

To place objects onto the worksheet from the Advanced Schematic Editor clipboard,

Choose Paste from the Edit menu,

The Edit Copy and Cut commands can be used to copy selected objects to the Schematic Editor clipboard.

See also

Paste

Save (OrCAD)

To copy objects to the Advanced Schematic Editor clipboard,

Choose Copy from the Edit menu.

The Edit Cut command can also be used to copy selected objects to the Schematic Editor clipboard.

See also

Copy

Import (OrCAD)

The Edit Copy and Paste commands are used to copy and place sections of a schematic worksheet into any open schematic document window.

See also

Copy
Paste

Export (OrCAD)

To save a block or group of objects to a file. Paste a selection of objects into a blank document window, then use the File Save As command to save the document to a new schematic file name.

See also

Paste

File Save AS

ASCII Import (OrCAD)

To import text into the schematic worksheet from an ASCII file, copy text to the Windows clipboard from another application (such as, Notepad or Write) and then paste the text into the text editing window of a Text Frame.

See also

Paste
Place Text Frame

Text Export (OrCAD)

To export text into an ASCII file, copy text to the clipboard from the text editing window of a Text Frame and then paste the text into another Windows application (such as, Notepad or Write). Once in the text editing application it can be saved as an ASCII text file.

See also

Text Frame
Copy

Conditions (OrCAD)

To display information about your computer system,

Choose Status from the Info menu.

Additional inforamtion about the current schematic worksheet can be obtained by using the Info Schematic and Info Components menu commands.

See also

Information System Status

Object (OrCAD)

To delete objects from your schematic worksheet,

Choose Delete from the Edit menu.

The Delete Single Object command can also be used to delete an individual object that has the focus.   All deletions can be restored by using the Edit Undo command (ALT+BACKSPACE).

See also

Delete
Delete Single Object

Delete (OrCAD)

Object
Block
Undo

Delete Block (OrCAD)

To delete groups of objects from your schematic worksheet,

Choose Clear from the Edit menu.

The Edit Cut command can also be used to remove selected objects from the worksheet and copies the selection to the Schematic Editor clipboard.

See also

Clear

Undo (OrCAD)

To restore deleted objects back onto the schematic worksheet,

Choose Undo from the Edit menu.

Multiple levels of undo are supported. The Edit Redo command can be used to restore changes made by the Undo command.

See also

Undo

Edit (OrCAD)

- **Edit**
- **Find**
- **Jump**
- **Zoom**

Edit (OrCAD)

To edit the attribues of an object on the schematic worksheet,

Choose Change from the Edit menu.

You can also position the cursor over the target object and double click LEFT MOUSE, or simply use the mouse to graphically change the object by by physically moving the position of the objects handles.

See also

Change

Find (OrCAD)

Equivalent options can be found in the Edit Search For menu. Such as searching for a text string, net label, part or a pin on the current worksheet.

See also

Find Text
Find And Replace Text
Find Next Text
Search for Net
Search for Part

Jump (OrCAD)

Equivalent options can be found in the Edit Jump menu. Such as jumping to a location mark, new location or to the origin of the current worksheet.

See also

Jump Location Mark
Jump New Location
Jump Origin
Jump Net
Find Text
Find And Replace Text
Find Next Text

Zoom (OrCAD)

Equivalent options can be found in the Zoom menu. Such as zooming by a defined area (zoom window, zoom point), magnification level (50%-400%), viewing the whole sheet (zoom sheet), viewing only the objects placed on the sheet (zoom all), zooming in and out on objects (zoom in, zoom out), redrawing the current document window (zoom redraw) and redrawing the current document window with the previous cursor location at the center of the window (zoom pan).

See also

Zoom Window
Zoom Point
Zoom 50%
Zoom 100%
Zoom 200%
Zoom 400%
Zoom Sheet
Zoom All
Zoom In
Zoom Out
Screen Redraw
Zoom Pan

Find (OrCAD)

Equivalent options can be found in the Edit Search For menu such as searching for a text string or net label. Parts (and their pins) and found by using the Jump button in the Library Browser.

See also

Find Text
Find And Replace Text
Find Next Text
Search for Net

Get (OrCAD)

To place parts on the schematic worksheet:

Choose Parts from the Place menu.

You can also place parts by pressing the Place button on the Component Browser.

See also

Place Parts
Place Part from Browser

HardCopy (OrCAD)

To print or plot a schematic worksheet:

Choose Print from the File menu.

The Setup Printer option in the File menu is used to select a printer or plotter

model, control printing options such as paper size, orientation, quality etc. and is also used to setup the scale, color mode and margins of your schematic sheet.

See also

File Print
Setup Printer

Inquire (OrCAD)

To view/change the text and name attributes associated with a probe, test vector index, stimulus, PCB layout and error marker objects,

Choose Change from the Edit menu.

You can also position the cursor over the target object and double click the LEFT MOUSE to view/change the text and name attributes.

See also

Change Items

A - H Tag (OrCAD)

To jump to a specified location on the schematic worksheet,

Choose one of the Location Mark options from the Jump menu.

Advanced Schematic supports up to 10 location marks, they are set by using the Location Marks command in the Place Menu.

See also

Jump Location Marks
Set Location Marks

Reference (OrCAD)

To move the cursor to a specified border reference or cell on the schematic worksheet, There is no equivilent command in Advanced Schematic.

X Location (OrCAD)

To move the cursor to a specified X location on the schematic worksheet,

Choose Jump Location from the Edit menu.

This command allows you to specify both an X and Y coordinate.

See also

Jump New Location

Y Location (OrCAD)

To move the cursor to a specified Y location on the schematic worksheet,

Choose Jump Location from the Edit menu.

This command allows you to specify both an X and Y coordinate.

See also

Jump New Location

Library (OrCAD)

Directory
Browse

Directory (OrCAD)

In Advanced Schematic, various methods are provided for reporting the components in sheets, projects and libraries.For example, all components in the current sheet (or project) are listed in the Part Designators list box in the Component Browser panel. The designator and part type are listed.

The File Reports Cross Reference command (shortcut: F, R, C) generates a listing of part types and designator labels, and the sheet location (filename) for each item. The report is generated for the current active sheet, hierarchical or multi-sheet flat project. This report is output in ASCII text format.

Detailed listings of sheet or project components are generated by the File Reports Bill of Materials command. When you generate this report, the Notepad utility is used to display the results on-screen immediately.

Bill of Materials report
Cross Reference report

Browse (OrCAD)

To browse and view the componets in a library file,

Choose Run Library Editor from the Library menu.

The Run Library Editor command is used to switch to the Schematic Library Editor. You can use the Main Panel or Component menu in the Library Editor to browse through the components in a library. Refer to the Schematic Library Editor reference manual and on-line help for further information.

See also

Run Library Editor

Macro (OrCAD)

Advanced Schematic supports the assignment of Hot Keys not keyboard Macros. Hot Keys allows you to assign keyboard keys and mouse button clicks to all of Advanced Schematic's command processes.

Choose Hot Keys from the Options menu.

A number of macro editing applications are available for recording keyboard

and mouse commands. The Windows Recorder for example, which is supplied with your Windows application can be used to record your keystrokes and mouse actions, and then play them back later by just pressing a key.

See also

Hot Keys

Place (OrCAD)

- **Wire**
- **Bus**
- **Junction**
- **Entry (Bus)**
- **Labels**
- **Module Port**
- **Power Object**
- **Sheet Symbols**
- **Text**
- **Dashed Line**
- **Trace**
- **Test Vectors**
- **Stimulus**
- **NoConnect**
- **Layout Directive**

Wire (OrCAD)

To place wires on the schematic worksheet,

Choose Wire from the Place menu.

While placing the wire, you can popup the Search For, Jump or Zoom menus by pressing the default hotkeys R, J, or Z.

See also

Place Wire

Bus (OrCAD)

To place bus lines on the schematic worksheet,

Choose Bus from the Place menu.

While placing the bus line, you can popup the Search For, Jump or Zoom menus by pressing the default hotkeys R, J, or Z.

See also

Place Bus

Junction (OrCAD)

To place junctions on the schematic worksheet,

Choose Junction from the Place menu.

While placing the junction, you can popup the Search For, Jump or Zoom menus by pressing the default hotkeys R, J, or Z.

See also

Place Junctions

Entry (OrCAD bus)

To place a bus entry on the schematic worksheet,

Choose Bus Entry from the Place menu.

While placing the bus entry, you can change the placement angle by pressing the SPACEBAR. You can also popup the Search For, Jump or Zoom menus by pressing the default hotkeys R, J, or Z.

See also

Place Bus Entry

Label (OrCAD)

To place labels on the schematic worksheet,

Choose Net Label from the Place menu.

While placing the net label, press the TAB key to manually change the default orientation, net name and size of the net label by typing new values directly into a dialog box.

You can also popup the Search For, Jump or Zoom menus by pressing the default hotkeys R, J, or Z.

See also

Place Net Label

Module Port (OrCAD)

To place a Module Port on the schematic worksheet,

Choose Port from the Place menu.

While placing the port, press the TAB key to manually change the default name, type and style of the port by typing new values directly into a dialog box.

You can also popup the Search For, Jump or Zoom menus by pressing the default hotkeys R, J, or Z.

See also

Place Port

Power (OrCAD)

To place power objects on the schematic worksheet,

Choose Power Port from the Place menu.

While placing the power port, press the TAB key to manually change the default orientation, value and type of the power port by typing new values directly into a dialog box.

You can also popup the Search For, Jump or Zoom menus by pressing the default hotkeys R, J, or Z.

See also

Place Power Port

Sheet Symbol (OrCAD)

To place a sheet symbol onto the schematic worksheet,

Choose Sheet Symbol from the Place menu.

Sheet nets are placed in the sheet symbol by using the Add Sheet Entry command. Use the Delete command from the Edit menu to delete a sheet entry. The name and file name of the sheet symbol can be changed by double clicking on them with the mouse or using the Change command from the Edit menu.

See also

Place Sheet Symbol
Add Sheet Entry

Text (OrCAD)

To place comments on the schematic worksheet,

Choose Text and then Annotation from the Place menu.

While placing the annotation, press the TAB key to manually change the default orientation, text and size of the net label by typing new values directly into a dialog box.

You can also popup the Search For, Jump or Zoom menus by pressing the default hotkeys R, J, or Z.

See also

Place Annotation

Dashed Line (OrCAD)

To place a dashed line on the schematic worksheet,

Choose Drawing Tools and then Line from the Edit menu.

While placing the line, press the TAB key to manually change the default width and style of the line by typing new values directly into a dialog box. There are three selectable line styles; solid, dashed and dotted. The dashed and dotted styles only apply to 'smallest' width lines.

You can also popup the Search For, Jump or Zoom menus by pressing the default hotkeys R, J, or Z.

See also

Place Line (Polylines)

Trace (OrCAD)

To place a trace on the schematic worksheet,

Choose Directives and then Probe from the Place menu.

While placing the probe, press the TAB key to manually change the default values of the probe by typing new values directly into a dialog box.

You can also popup the Search For, Jump or Zoom menus by pressing the default hotkeys R, J, or Z.

See also

Place Probe

Test Vectors (OrCAD)

To place a vector on the schematic worksheet,

Choose Directives and then Test Vector Index from the Place menu.

While placing the Test Vector Index, press the TABS key to manually change the default values by typing new values directly into a dialog box.

You can also popup the Search For, Jump or Zoom menus by pressing the default hotkeys R, J, or Z.

See also

Place Test Vector Index

Stimuli (OrCAD)

To place a stimulus on the schematic worksheet,

Choose Directives and then Stimulus from the Place menu.

While placing the stimulus, press the TAB key to manually change the default values by typing new values directly into a dialog box.

You can also popup the Search For, Jump or Zoom menus by pressing the default hotkeys R, J, or Z.

See also

Place Stimulus

NoConnect (OrCAD)

To place a no-connect symbol on the schematic worksheet,

Choose Directives and then No ERC from the Place menu.

While placing the No ERC symbol, you can also popup the Search For, Jump or Zoom menus by pressing the default hotkeys R, J, or Z.

See also

Place No ERC

Layout Directive (OrCAD)

To place a layout directive on the schematic worksheet,

Choose Directives and then PCB Layout from the Place menu.

While placing the PCB Layout, press the TAB key to manually change the default values by typing new values directly into a dialog box.

You can also popup the Search For, Jump or Zoom menus by pressing the default hotkeys R, J, or Z.

See also

Place PCB Layout

Quit (OrCAD)

Enter Sheet (OrCAD)

To view a schematic worksheet represented by a sheet symbol in a hierarchical design,

Choose either Up Hierarchy or Down Hierarchy from the File menu.

When moving down the hierarchy, if the child sheet is not already opened Advanced Schematic will automatically open the sheet (any number of schematic worksheet files can be opened at the same time). To move up the hierarchy, the parent sheet needs to be already open.

You can also use the Project Manager to visually navigate through the schematic worksheets in a project.

See also

Up Hierarchy
Down Hierarchy
Toggle Project Manager

Leave Sheet (OrCAD)

To move up one level in the schematic hierarchy,

Choose Up Hierarchy from the File menu.

To move up the hierarchy, the parent sheet needs to be already open. You can also use the Project Manager to visually navigate through the schematic worksheets in a project.

See also

Up Hierarchy
Toggle Project Manager

Update File (OrCAD)

To save the changes made to the current schematic worksheet,

Choose Save from the File menu.

If the schematic worksheet has not been previously saved, the Save File As dialog box will open, prompting for a file name.

See also

File Save

Write To File (OrCAD)

To save the current schematic worksheet to a new path, file name or format.

Choose Save As from the File menu.

See also

File Save As

Initialize (OrCAD)

To load a schematic worksheet file,

Choose Open Sheet from the File menu.

To create a new blank schematic worksheet,

Choose New from the File menu.

See also

File Open Sheet
File Open Project
File New

Suspend To System (OrCAD)

Use the Windows Program Manager to switch to the DOS operating system.

1. While you are in the Advanced Schematic Editor press the ALT+TAB keys to switch to the Program Manager,
2. Click on the DOS icon (usually stored in the Main program group), or
3. Choose Run from the File menu and type *Command*, then click OK or press ENTER.

Abandon Edits (OrCAD)

To quit the current session of Advanced Schematic Editor,

Choose Exit from the File menu.

If any schematic worksheet has been modified since last saving, a dialog box will be displayed that prompts to save the file(s).

See also

File Exit

Run User Commands (OrCAD)

Use the Windows Program Manager to run external DOS commands.

1. While you are in the Advanced Schematic Editor press the ALT+TAB keys to switch to the Program Manager,
2. Choose Run from the File menu and type in the DOS command, then click OK or press ENTER.

Repeat (OrCAD)

To duplicate the last placed object, or to place an array of objects on the worksheet:

First cut or copy the object(s) to be repeated to the clipboard, then choose Paste or Paste Array from the Edit menu.

See also

Paste Array
Execute Paste Array

Set (OrCAD)

Backup File (OrCAD)

A backup file with the extension of .BAK, is created every time you save a schematic worksheet or project.

Advanced Schematic also supports the creation of additional backup files during an editing session. In the Preferences dialog box, you can set the interval for generating an automatic backup file and set the number of automatic backup files used in a rotating file system.

See also

Drag Buses (OrCAD)

To enable/disable dragging of bus and wire segments,

Choose Preferences from the Options menu and enable/disable the drag option.

See also

Preferences

Error Bell (OrCAD)

To turn the Windows warning beep and system sounds on or off,

Choose the Sound icon from the Windows control panel and click on the Enable System Sounds check box.

Left Button (OrCAD)

Advanced Schematic Editor does not have an equivalent to the Left Button command.

Macro Prompts (OrCAD)

Advanced Schematic Editor does not have an equivalent to the Macro Prompts command.

Orthogonal (OrCAD)

While you are placing wire or bus segments you can toggle through the placement angle modes by pressing the SPACEBAR. Wires and buses can be drawn at 90, 45 and any angle degrees.

See also

Place Wire
Place Bus

Show Pins (OrCAD)

To show/hide pin numbers,

Choose Preferences from the Options menu and click on the Pin Numbers check box.

See also

Preferences

Title Block (OrCAD)

To turn the Advanced Schematic (Standard or ANSI) title block on or off,

Choose Sheet from the Options menu and click on the Title Block check box.

See also

Sheet Options

Worksheet Size (OrCAD)

To select a schematic worksheet size,

Choose Sheet from the Options menu and select a sheet size.

Advanced Schematic supports 10 standard imperial and metric sheet sizes. They include A, B, C, D, E (or metric sizes A4-A0). You can also define your own custom sheet size up to 65 inches square.

See also

Sheet Options

X,Y Display (OrCAD)

To turn the current cursor position on or off, you need to turn the whole status line on or off,

Choose Status from the Options menu.

The left side of the status line indicates the current cursor position (the origin is the bottom left corner of the worksheet). The status line also displays information about the current activity or mode.

See also

ToggleStatus line

Grid Parameters (OrCAD)

Grid References

To turn the worksheet border on or off, choose Sheet from the Options menu and click on the Border check box.

Stay on Grid

To turn the snap grid on or off, choose Snap Grid from the Options menu.

Visible Grid Dots

To turn the visible grid on or off, choose Sheet from the Options menu and click on the Visible Grid check box. You can also choose Visible Grid from the Options menu.

See also

Sheet Options
Toggle Snap Grid
Toggle Visible Grid

Repeat Parameters (OrCAD)

To set up the paste array parameters,

Choose Paste Array from the Edit menu.

In the Paste Array dialog box the Horizontal option refers to X Repeat Step, Vertical refers to Y Repeat Step and Text Increment refers to Label Repeat Delta.

There is an option in the Preferences dialog box to turn on/off text and number incrementing.

See also

Paste Array
Preferences

Visible Lettering (OrCAD)

Advanced Schematic Editor does not have an equivalent to the Visible Lettering commands.

Tag (OrCAD)

To tag a specified location on the schematic worksheet,

Choose one of the Location Mark options from the Place menu.

Advanced Schematic supports up to 10 location marks. Use the Jump Location Mark options from the Edit menu to move the cursor to these set locations.

See also

Set Location Marks
Jump Location Marks

Center (OrCAD)

To center the worksheet view around the current cursor location,

Choose Pan from the Zoom menu.

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

See also

Zoom Pan

Zoom (OrCAD)

- **Center**
- **In**
- **Out**
- **Select**

In (OrCAD)

To zoom in on the schematic worksheet,

Choose In from the Zoom menu.

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

See also

Zoom In

Out (OrCAD)

To zoom out, showing more of the schematic worksheet,

Choose Out from the Zoom menu.

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

See also

Zoom Out

Select (OrCAD)

To select a zoom scale,

Choose one of the four zoom scales from the Zoom menu.

See also

Zoom 100%
Zoom 200%
Zoom 100%
Zoom 50%

Printing

Screen Regions

Reference

Protel Schematic 3 (DOS) Help

- Block
- Current
- Delete
- Edit
- File
- Highlight
- Information
- Jump
- Library
- Move
- Place
- Repeat
- Setup
- Undelete
- Zoom

Copy (Schematic 3)

To make a copy of the objects that are currently selected,

Choose Copy and then Paste from the Edit menu.

Make sure that the current selection includes only those objects you wish to copy. You can use the Edit Select and De-Select menu commands to select and de-select objects.

Copying a selection

See also

Copy

Paste

Block (Schematic 3)

Define (Schematic 3)

There are several ways to select a group of objects,

1. Use the Select and Toggle Selection commands from the Edit menu. The Select Inside Area command is equivalent to Block Define.
2. Use SHIFT+LEFT MOUSE to add individual objects to the current selection group,
3. Enable the objects' selection field within its dialog box, it is also possible to use global editing commands to select multiple objects.

Advanced Schematic allows more than one block or group of objects to be selected at any one time. As you select objects they are added to the current selection group.

See also

Select Inside Area
Toggle Selection
Toggle Single Object

Hide (Schematic 3)

There are several ways to de-select objects,

1. Use the De-select and Toggle Selection commands from the Edit menu. The De-select All command is equivalent to Block Hide.
2. Use SHIFT+LEFT MOUSE to remove individual objects from the current selection group,
3. Disable the objects' selection field within its dialog box, it is also possible to use global editing commands to de-select multiple objects.

See also

De-Select Inside Area
Toggle Selection
Toggle Single Object

Inside Delete (Schematic 3)

To delete all the selected objects from the schematic worksheet,

Choose Cut or Clear from the Edit menu.

Make sure that the current selection includes only those objects you wish to delete. You can use the Edit Select and De-Select menu commands to select and de-select objects.

See also

Cut

Clear

Move (Schematic 3)

To move selected objects,

Choose Move and then Move Selection from the Edit menu.

Make sure that the current selection includes only those objects you wish to move. You can use the Edit Select and De-Select menu commands to select and de-select objects.

See also

Move Selection

Outside Delete (Schematic 3)

To delete objects outside a defined area,

Choose Select Outside Area to select the objects to be deleted and then choose Cut or Clear from the Edit menu.

See also

Select Outside Area
Cut
Clear

Read (Schematic 3)

The Copy and Paste commands from the Edit menu are used to copy and place sections of a schematic worksheet into any open schematic document window.

See also

Copy

Paste

Write (Schematic 3)

To save a block or group of objects to a file. Paste a selection of objects into a blank document window, then use the File Save As command to save the document to a new schematic file name.

See also

Paste
File Save As

Current (Schematic 3)

There is no equivilent command in Advanced Schematic. The current default settings for most items changes with the last placement of the item.

Line Type
Sheet Size
Junction Size

Line Type (Schematic 3)

Advanced Schematic Editor supports three different types of objects, wires, buses and lines.

Wires are used instead of Thin Signal and Thick Signal lines. There are four wire sizes, smallest, small, medium and large.

Buses are used instead of the Bus line. There are four bus sizes, smallest, small,

medium and large.

Lines are used instead of the Dashed line. Lines have three selectable line styles, solid, dashed and dotted. The dashed and dotted styles only apply to 'smallest' width lines.

See also

Place Wire
PlaceBuss
Place Line (Polylines)

Sheet Size (Schematic 3)

To select a schematic worksheet size,

Choose Sheet from the Options menu and select a sheet size.

Advanced Schematic supports 10 standard imperial and metric sheet sizes. They include A, B, C, D, E (or metric sizes A4-A0). You can also define your own custom sheet size up to 65 inches square.

See also

Sheet Options

Junction Size (Schematic 3)

Advanced Schematic Editor supports four junction sizes, smallest, small,medium and large.

While placing a junction, press the TAB key to manually change the default junction size. When using the Place Junction command the default size is determined by the last placed junction.

See also

Place Junctions

Delete (Schematic 3)

To delete objects from your schematic worksheet,

> Choose Delete from the Edit menu, you do not need to specify the type of object to be deleted, (e.g. Delete Junction).

The Delete Single Object command can also be used to delete an individual object that has the focus. All deletions can be restored by using the Edit Undo command (ALT+BACKSPACE).

See also

Delete
Delete Single Object

Utilities (OrCAD)

- ANNOTATE.EXE
- DRAFT.EXE
- LIBLIST.EXE
- LIBARCH.EXE
- BACKANNO.EXE
- COMPOSER.EXE
- TREELIST.EXE
- PARTLIST.EXE
- HFORM.EXE
- LIBEDIT.EXE
- PRINTALL.EXE

ANNOTATE.EXE (OrCAD)

To rename all the Parts on a schematic worksheet or project.

Choose Annotate from the File menu.

See also

Annotate

DRAFT.EXE (OrCAD)

Advanced Schematic Editor Application.

LIBLIST.EXE (OrCAD)

To add and remove libraries from the library list,

Choose Add/Remove from the Library menu, or click the Add/Remove button in the Component Browser panel.

An unlimited number of libraries may be added to this list. Advanced Schematic only stores the names of the library components into memory.

See also

Add/Remove Library

LIBARCH.EXE (OrCAD)

To make a component library of all the parts in a schematic project,

Choose Make Project Library from the Library menu.

See also

Make Project Library

BACKANNO.EXE (OrCAD)

To back annotate part and pin information from a PCB was/is File,

Choose Back Annotate from the File menu.

See also

Back Annotate

COMPOSER.EXE, DECOMP.EXE (OrCAD)

To compile and decompile library files, use the Open and Save As commands from the Schematic Library Editor File menu.

When using the Open command, Schematic Library Editor automatically recognises OrCAD library source files and Advanced Schematic library text files.

When using the Save As command, simply specify the type of output format for the Library file.

TREELIST.EXE (OrCAD)

To generate a text listing to represent the design hierarchy of a schematic project,

Choose Reports and then Project Hierarchy from the File menu.

See also

Report Project Hierarchy

PARTLIST.EXE (OrCAD)

To generate a bill of materials file,

Choose Reports and then Bill of Material from the File menu.

See also

Bill of Materials

HFORM.EXE, IFORM.EXE (OrCAD)

To create a hierarchical or flat netlist file,

Choose Create Netlist from the File menu.

Net Check Report

To perform an Electrical Rules Check on the current project,

Choose Reports and then Electrical Rules Check from the File menu.

Two other report files can be generated from the File Reports menu, a cross reference file and a project hierarchy file.

See also

Create Netlist

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.

See also

Run Library Editor

PRINTALL/ PLOTALL.EXE (OrCAD)

To print or plot a schematic worksheet,

Choose Print from the File menu.

The Setup Printer option in the File menu is used to select a printer or plotter

model, control printing options such as paper size, orientation, quality etc., and is also used to setup the scale, color mode and margins of your schematic sheet.

See also

File Print
Setup Printer

Edit (Schematic 3)

To edit the attribues of an object on the schematic worksheet,

> Choose Change from the Edit menu, you do not need to specify the type of object to be changed, (i.e. Change Junction).

You can also position the cursor over the target object and double click the LEFT MOUSE, or simply use the mouse to graphically change the object by physically moving the position of the objects handles.

Part pins can not be edited or moved within the Schematic Editor. Pin changes must be made at library level, using the Schematic Library Editor. Please refer to the Schematic User Guide, for detailed explanations regarding library and component management concepts.

To edit the text within a text frame, click the change button in the text frames Change dialog box.

See also

Change

File (Schematic 3)

- Clear
- Dos
- Files
- Load
- Path
- Quit
- Save

Clear (Schematic 3)

To create a new blank schematic worksheet,

Choose New from the File menu.

See also

File New

DOS (Schematic 3)

Use the Windows Program Manager to switch to the DOS operating system.

1. While you are in the Advanced Schematic Editor press the ALT+TAB keys to switch to the Program Manager,
2. Click on the DOS icon (usually stored in the Main program group), or
3. Choose Run from the File menu and type COMMAND, then click OK or press ENTER.

Files (Schematic 3)

You can use the Open Sheet or Save As commands from the File menu to display a directory listing of files.   The Windows File Manager can also be used to view and organize your files and directories.

Load (Schematic 3)

To load a schematic worksheet file,

Choose Open Sheet from the File menu.

The Open Project command is used to load and display the contents of all the hierarchical worksheet files contained in a schematic project.

Converting Protel 3 libraries

See also

File Open Sheet
File Open Project

Path (Schematic 3)

Use the Windows Program Manager to set a new working directory for your worksheet files.

1. Select the Schematic Editor icon,
2. Choose Properties from the Program Manager File menu,
3. Change the working directory in the Program Item Properties dialog box.

Quit (Schematic 3)

To quit the current session of Advanced Schematic Editor,

Choose Exit from the File menu.

If any schematic worksheet has been modified since last saving, a dialog box will be displayed that prompts to save the file(s).

See also

File Exit

Save (Schematic 3)

To save the changes made to the current schematic worksheet,

Choose Save from the File menu.

To save the current schematic worksheet to a new path, file name or format.

Choose Save As from the File menu.

See also

File Save
File Save As

Highlight (Schematic 3)

To select all the connections making up a Net,

Choose Select and then Net from the Edit menu.

The Select Connection command can be used to select the wires, junctions and net labels that are connected between two pins.

See also

Components, listing sheet (Schematic 3)

To display a list of Parts on the current worksheet,

Choose the File Reports Bill of Materials command or File Reports Cross Reference command. These reports will be quickly generated for the current file/project and will be opened using the Notepad utility. You can also map these report text files to other text editors, using the File Run options.

Report BOM process
Report Cross Reference process

See also
Setup User Programs process

Information (Schematic 3)

Components
Highlighted Pins
Library
Status

Highlighted Pins (Schematic 3)

To display a list of the pins that are currently selected on the worksheet,

Choose Selected Pins from the Info menu.

The listing can be saved as a text file and can be sent directly to the printer port by entering the DOS reserved name PRN when you are prompted to supply a file name.

You can use the Schematic Library Editor to display a complete listing of the contents of a library file. Refer to the Schematic Library Editor reference manual and on-line help for further information.

See also

Information Selected Pins

Information (Schematic 3 library)

The Schematic Library Editor can be used to display a complete listing of the contents of a library file. Refer to the Schematic Library Editor reference manual and on-line help for further information.

Status (Schematic 3)

To display information about your computer system,

Choose Status from the Info menu.

See also

Information System Status

Jump (Schematic 3)

- Component
- Location
- Net
- Origin
- Text

Jump Component (Schematic 3)

To jump to a component on the current worksheet,

Choose Search for and then Part from the Edit menu,

When the Component Browser is displayed, you can jump to a part within the current Project by first highlighting a component with the selection bar in the Components list box, then pressing the Jump To button.

See also

Toggle Component Browser process
Jump to Part process

Location (Schematic 3)

To move the cursor to a specified location,

Choose Jump and then New Location from the Edit menu.

See also

Jump New Location

Net (Schematic 3)

To move the cursor to a specified Net,

Choose Search for and then Net from the Edit menu.

See also

Search for Net

Origin (Schematic 3)

To move the cursor to the lower left corner of the schematic worksheet,

Choose Jump and then Origin from the Edit menu.

See also

Jump Origin

Text (Schematic 3)

To search for text on the schematic worksheet,

Choose Search for and then String from the Edit menu.

See also

Find Text
Find And Replace Text
Find Next Text

Library (Schematic 3)

- **Browse**
- **Components**
- **Load**

Components, listing library (Schematic 3)

Components in the current sheet are listed (Reference designator and name)in the Components box in the Browse window. You can quickly generate a list of components by choosing either File Reports Bill of Materials or File Reports Cross Reference. These options generate text files which list components. The reports are opened as Notepad documents when either command is selected.

Report BOM process
Report Cross Reference process

See also
(Toggle) Component Browser process

Load (Schematic 3)

To add and remove libraries from the library list,

Choose Add/Remove from the Library menu, or click the Add/Remove button in the Component Browser panel.

An unlimited number of libraries may be added to this list. Advanced Schematic only stores the names of the library components into memory.

See also

Add/Remove Library

Move (Schematic 3)

To move objects around the schematic worksheet,

Choose Move and then Move from the Edit menu.

Wires will retain their orthogonality as objects are moved around the worksheet. You can also use the shortcut 16 CTRL + LEFT MOUSE to pick up and move an object.

The Change Object Graphically command can be used to move and clean up individual bus, wire and line segments. There is not a specific menu command for this process.

While moving wires or bus segments you can use the SPACEBAR to toggle through the different angle modes of wire and bus placement. The INSERT and DELETE keys can be used to add and remove line segments.

See also

Move
Change Object Graphically

Place (Schematic 3)

Annotation (Schematic 3)

To place comments on the schematic worksheet,

Choose Text and then Annotation from the Place menu.

While placing the annotation, press the TAB key to manually change the default orientation, text and size of the annotation by typing new values directly into a dialog box.

See also

Place Annotation

Component, placing (Schematic 3)

To place components on the schematic worksheet,

Choose Parts from the Place menu or click the Component tool button on the Wiring tools palette.

You can also place parts by clicking the PLACE button on the Component Browser.

See also

Place Parts
Place Part from Browser

Documentation Box (Schematic 3)

To place detailed descriptions and comments on the schematic worksheet,

Choose Text and then Text Frame from the Place menu.

While placing the text frame, press the TAB key to manually change the

default text, font style, text color, border width etc., by typing new values

directly into a dialog box.

Text Frames support the cut, copy and paste commands for text editing. It is possible to copy text to the Windows clipboard from another application and then paste the text into the text editing window.

See also

Place Text Frames

Junction (Schematic 3)

To place junctions on the schematic worksheet,

Choose Junction from the Place menu.

While placing the junction, press the TAB key to manually change the default junction size and color by typing new values directly into a dialog box.

See also

Place Junctions

Line (Schematic 3)

Advanced Schematic Editor supports wires, buses and lines as separate objects.

Wires are used instead of Thin Signal and Thick Signal lines. There are four wire sizes, smallest, small, medium and large.

Buses are used instead of the Bus line. There are four bus sizes, smallest, small, medium and large.

Lines are used instead of the Dashed line. Lines have three selectable line styles, solid, dashed and dotted. The dashed and dotted styles only apply to smallest width lines.

See also

Place Wire
Place Bus
Place Line (Polylines)

Net Label (Schematic 3)

To place net labels on the schematic worksheet,

Choose Net Label from the Place menu.

While placing the net label, press the TAB key to manually change the default orientation, net name and size of the net label by typing new values directly into a dialog box.

See also

Place Net Label

Repeat (Schematic 3)

To set up the repeat array parameters,

Choose Paste Array from the Edit menu.

In the Paste Array dialog box the Place button and the Execute Paste Array command

refers to the Execute Repeat command, Item Count refers to Default Count, Text Increment refers to Component Increment, Horizontal and Vertical option refers to the X and Y Offsets.

See also

Paste Array
Execute Paste Array

Setup (Schematic 3)

Keys (Schematic 3)

Advanced Schematic supports the assignment of Hot Keys not keyboard Macros. Hot Keys allows you to assign keyboard keys and mouse button clicks to all of Advanced Schematic Editor command processes.

Choose Hot Keys from the Options menu.

A number of macro editing applications are available for recording keyboard and mouse commands. The Windows Recorder for example, which is supplied with your Windows application can be used to record your keystrokes and mouse actions, and then play them back later by just pressing a key.

See also

Hot Keys

Menu Colors (Schematic 3)

You can use the Windows Program Manager to change the color of Windows screen elements. Such as menu text, menu bar, window frame, window background, title bar text, buttons, etc.

Choose the Color icon from the Windows control panel and select a new color scheme or use the color palette to customize your own.

Options (Schematic 3)

Auto Pan

Choose Preferences from the Options menu and click on the Auto Pan check box.

Backup

In the Preferences dialog box, you can set the interval for generating an automatic backup file and set the number of automatic backup files used in a rotating file system.

Cursor Type

In the Preferences dialog box, three options are available for the physical (or sheet) cursor, a large 90 degree cross, small 90 degree cross or a 45 degree cross. The sheet cursor is displayed when making selections, moving or placing objects.

Drag

Choose Preferences from the Options menu and enable/disable the drag option.

Ortho Mode

While you are placing wire or bus segments you can toggle through the placement angle modes by pressing the SPACEBAR. Wires and buses can be drawn at 90, 45 and any angle degrees.

Title Block

Choose Sheet from the Options menu and click on the Title Block check box.

Sheet Border

Choose Sheet from the Options menu and click on the Border check box.

View Device Pins

Device Suffix

Question Delete

Choose Preferences from the Options menu and click on the Question Delete check box.

Flash Cursor

Advanced Schematic Editor does not support the Flash Cursor command. You can use the Desktop settings from within the Windows Control Panel to change the blink rate of the text cursor. The text cursor is used for text input windows, such as, name fields or the text editing window of text

See also

Preferences
Sheet Options

Display Colors (Schematic 3)

BackGround

Choose Sheet from the Options menu and click in the Color, color box.

Sheet

Choose Sheet from the Options menu and click in the Border Color, color box.

Highlight

Choose Preferences from the Options menu and click in the Selections, color box.

Grid

Choose Preferences from the Options menu and click in the Grid Color, color box.

Objects

Each object primitive on the schematic worksheet can be assigned individual color attributes, color changes can also be applied globally. This allows you to color code wires, classes of parts or individual items. Many objects such as rectagles, ellipses and text frames have a separate color assignment for their outline and fill area.

To assign a new color to objects,

Click inside the color box within the objects change dialog box.

See also

Preferences
Sheet Options
Change

String Defaults (Schematic 3)

Advanced Schematic Editor does not have an equivalent to the String Defaults command. The default text size and style is determined by the last placed text string. Text sizes and styles can be globally changed across the schematic worksheet.

Undelete (Schematic 3)

To restore deleted objects back onto the schematic worksheet,

Choose Undo from the Edit menu.

Multiple levels of undo are supported. The Edit Redo command can be used to restore changes made by the Undo command.

See also

Undo

Zoom (Schematic 3)

- Redraw
- Pan
- Expand
- Contract
- Select
- All

Redraw (Schematic 3)

To redraw the objects on the current schematic worksheet.

Choose Redraw from the Zoom menu.

See also

Screen Redraw

Pan (Schematic 3)

To center the worksheet view around the current cursor location,

Choose Pan from the Zoom menu.

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

See also

Zoom Pan

Expand (Schematic 3)

To zoom in on the schematic worksheet,

Choose In from the Zoom menu.

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

See also

Zoom In

Contract (Schematic 3)

To zoom out, showing more of the schematic worksheet,

Choose Out from the Zoom menu.

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

See also

Zoom Out

Select (Schematic 3)

To select a zoom scale,

Choose one of the four zoom scales from the Zoom menu.

See also

Zoom 400%
Zoom 200%
Zoom 100%
Zoom 50%

All (Schematic 3)

To fit all objects on the current sheet in the window

Choose All from the Zoom menu.

See also

Zoom All

Utilities (Schematic 3)

- **ANNOTATE.EXE**
- **POST.EXE**
- **NETTRAN.EXE**
- **NETCHECK.EXE**
- **SLM.EXE**
- **SCHEDIT.EXE**
- **SCHPLOT.EXE**

ANNOTATE.EXE (Schematic 3)

To rename all the Parts on a schematic worksheet or project.

Choose Annotate from the File menu.

See also

Annotate

POST.EXE (Schematic 3)

Netlist

To generate a Protel netlist file,

Choose Create Netlist from the File menu, select Protel format.

Net Check Report

To perform an Electrical Rules Check on the current project,

Choose Reports and then Electrical Rules Check from the File menu.

BOM

To generate a bill of materials file,

Choose Reports and then Bill of Material from the File menu.

Two other report files can be generated from the File Reports menu, a cross reference file and a project hierarchy file.

See also

Create Netlist
Bill of Materials

NETTRAN.EXE (Schematic 3)

To generate a netlist file in another format, other than Protel.

Choose Create Netlist from the the File menu, select the desired output format.

Advanced Schematic Editor supports many more netlist output formats than NETTRAN.

See also

Create Netlist

NETCHECK.EXE (Schematic 3)

SLM.EXE (Schematic 3)

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.

See also

Run Library Editor

SCHEDIT.EXE (Schematic 3)

Advanced Schematic Editor Application.

SCHPLOT.EXE (Schematic 3)

To print or plot a schematic worksheet,

Choose Print from the File menu.

The Setup Printer option in the File menu is used to select a printer or plotter model, control printing options such as paper size, orientation, quality etc., and is also used to setup the scale, color mode and margins of your schematic sheet.

See also

File Print
Setup Printer

Data Primitives

Advanced Schematic Help

Reference

Concepts

Compatibility

Concepts

Presentation quality schematics

Protel for Windows Advanced Schematic provides extensive graphic capabilities for designing schematics. By using graphical features such as colors, TrueType fonts, and graphical images you can highlight, emphasize and illustrate important elements of the design. Advanced Schematic also supports high-resolution printing and plotting of schematic sheets, using a wide range of Windows-supported monochrome and color devices.

Using color

Using fonts

Image files

Printing

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

Batch Mode prints

When using this option from the Schematic Editor, this option prints either a single sheet or a batch of all open sheets (including all opened projects). When using this option from the Library Editor, prints a single component (from the window that is the current focus) or all components in the current library. The latter 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.

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.

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.

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.

Process (Hotkey)

Selects the process for hot key assignment.

Current Process (Hotkey)

This box lists the current process assigned to the selected key combination.

Description (Hotkey)

This box describes the purpose or use of the current process.

Assign (Hotkey)

Adds the key and process selections to the current hot key file (.KEY extension).

Load (Hotkey)

Loads an existing key file (.KEY extension) for editing.

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.

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.

Special objects

Error markers

Error markers are placed in schematic project sheets automatically

during ERC (Electrical Rules Check) generation. These objects cannot be placed manually. A 360 degree arc is placed at the site of each connective error or warning. The ERC error markers can be deleted from the sheet by clicking over the object with LEFT MOUSE and pressing DELETE.

Special strings

Special strings are special pre-defined single line text objects you place in sheets, which generate automatic output when a sheet is printed or plotted.

Special strings

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

Capture

Capture refers to the ability of schematic editors to export the connectivity in a circuit in the form of a netlist.

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.

Net (Select)

This command selects all wires and pins that connect a physical/logical net on the current sheet and nets on all other open sheets with the same net label. To use this feature:

Choose Edit Select Net;

You will be prompted "Select Net."

Position the cursor over a wire within the desired net and press ENTER or LEFT MOUSE.

The continuous net extending from the selected wire will be displayed in the selection color.

See also

Connection (Select)

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.

See also

Net (Select)

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.

OrCAD net identifiers

In OrCAD SDT, *ports* are called *module ports* and *sheet entries* are called *sheet nets*. The OrCAD model of hierarchy refers to horizontal intersheet connections, using module ports as a *flat design*.

Cursor shape

Options include: large 90 degree cross (cross hair type), small cross or 45 degree cross.

Question Delete option

When enabled, prompts the user to confirm deletion, before the deletion is completed.

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.

OrCAD Port Length

Advanced Schematic allows the user to set ports to any length. This option preserves a fixed length for ports, so that these items will be compatible with OrCAD SDT, which sets module port length based upon the number of text characters in the port's label. Choose this option if you intend to export the sheet in OrCAD format, otherwise the port may not be aligned with the connecting wire or bus, resulting in an error condition.

Drag Wires

When enabled, this option maintains wire-to-component pin and bus connections when components are moved. As you move components, the wires will drag (stretch) to maintain the connection. Within reason, orthogonality will be maintained -- that is, wires will keep their horizontal/vertical orientation as far as possible. If you rotate the components, change the grid or move the component beyond the reach of the orthogonality feature, some segments may "rubberband" to an "any angle" placement mode. Use the Move Break Wire command (or press CTRL+SHIFT while clicking on wire segments) to restore orthogonal placement.

Auto Backup

A backup version of your sheet file is created whenever you save a file or project. This is named with the extension *.BAK.

OrCAD project management

OrCAD SDT users will be familiar the concept of master sheets, although this system handles project management somewhat differently. OrCAD using special objects, called pipelines to link "flat" project sheets. Under Advanced Schematic, sheet symbols represent each sheet in all multi-sheet projects.

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.

NETCHECK.EXE (Schematic 3)

NETCHECK reads and compares the net (not component) information in two netlists.

To compare two netlists in Advanced Schematic, choose the File Reports Netlist Compare command.

Netlist Format notes

See also

Netlist Compare

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.

Place Port process
Place Net Labels process

Set Auto Pan (OrCAD)

Turns OrCAD Auto Pan off/on.

Preferences

Document

This option, in the Options Sheet dialog box, allows the user to specify information which is added to the sheet title block when the ANSI title block option is selected, including the organization name, address, sheet number, total sheets, title, number and revision. These will be ignored is the normal title block is selected.

Each of these fields can be defined as Special Strings. If you place a string, the text stored in its field in this dialog box will be printed with the sheet:

.organization

.address1

.address2

.address3

.address4

.sheetnumber

.sheettotal

.title

.documentnumber

.revision

Run command options

Tool menu commands 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

Browser

The Browse window provides search for and jump to functions for all primitives placed in all open sheets.

Browser panel

The Browser panel is provided as an aid in searching for, and placing component parts into sheets and for locating other objects during schematic editing. This panel can be displayed as needed or hidden, to free more of the display area for the sheet workspace.

Text button

Click the text button at the extreme bottom left of the Browser panel to change the text of the designator (for parts) or label (other objects with text). If the object does not have a text label, clicking the button is ignored.

Jump button

Click the Jump button to jump the the highlighted primitive name in the list box. The display will pan to center the object and the object will be placed in focus, displaying its graphical editing handles or control points, if any (see Focus, below).

Edit button

Click the Edit button to open the primitives dialog box, which provides access to all object attributes (see Changing objects, below).

Read Only button

The Read Only button allows users view the Read Only fields of a component. Use the Read Only button by selecting a component from the list above and then clicking the Read Only button. This action makes the display jump to the selected part and opens a dialog box with the part's Read Only Fields information.

Read Only Fields are created with a description a part in its library. These fields are not displayed on the schematic sheet nor are they printed.

Change button

The Change button allows users to directly access a listed part's dialog box. Scroll through the Components in Library list to choose a component, then click the Change button. This action makes the display jump to the selected part and opens the part's dialog box.

Browse Nets option

Selecting the object type "Net Identifiers" allows viewing all net identifiers in the current project, including net labels, ports, power objects, and sheet entries. Listings include the name of the net, type of object, sheet name and coordinates. Users can jump to or change Net objects with the Jump and Change buttons.

Report ERC process
Electrical Rules Check process

See also

Jump to Error command

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.

Global changes

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.

Change Part Dialog Box

This dialog box has changed considerably since the *Advanced Schematic Reference* was printed (see page 231 in the manual). The following additional fields are now part of this dialog box:

Lib Ref

This is the library component identifier. You can change the component from this box and make global changes, using the Wildcard:Wildcard and Change From/To fields.

Designator

This component field, the reference designator for the component on the sheet, is now editable directly from the Change box and can be used in global edits.

Part Type

This component field, normally used to display a value, is now editable directly from the Change box and can be used in global edits.

Footprint

Up to four footprints (or component patterns) can be assigned to components, using the Component Description command to access all component text fields.

Once a component is placed. The Change Component dialog box provides access to the footprint description fields. By default, the footprint pattern named in Footprint Field 1 (in the Component Description dialog) will be displayed.

Attributes

Up to 8 text strings (of up to 255 characters each) can be defined for parts, when they are played on the sheet. You can display these fields by enabling the Hidden Fields option. These fields can also be used when performing global edits.

These 8 part attribute fields are separate from the 8 text fields which can be stored for a component in the library. Library text fields cannot be displayed in the sheet, but are included in CSV format Bill of Materials reports.

Netlist format notes

Protel

Part Types can contain up to 12 characters

Footprint descriptions can be up to 12 characters

Part designators can be up to 12 characters

Net Names can contain up to 20 characters

Pin Numbers are limited to 4 characters

Wirelist

Footprints can contain up to 29 characters

Designators can contain up to 9 characters

Pin Numbers are limited to 7 characters

Pin Names are limited to 15 characters.

Tango

Part Types can be up to 16 characters in length

Foot Prints can be up to 16 characters in length

Designators can be up to 16 characters in length

Net Names can be up to 16 characters in length

Designator and ModuleName must be uppercase only

Cadnetix

Part Types can contain up to 17 characters

Module Names can contain up to 15 characters

Designators plus Pin Numbers can contain up to 12 characters

Net Names can contain up to 16 characters

Node Numbers are not checked for length

Pin Numbers can contain up to 3 digits

Calay

Part Names can contain up to 19 characters

Module Names can contain up to 19 characters

Reference Strings can contain up to 19 characters

Node Names can contain up to 8 characters (see legal characters below)

Node Numbers are limited to 5 digits (plus the leading 'N')

With the Calay netlist format, components are written into a separate file, named <project filename>.PRT. Nets will be written into <project filename>.NET.

CBDS

Net Names can contain up to 20 characters

ComputerVision

Net Names are limited to 18 characters

EEDesigner

Designators are limited to 8 characters

Net Names are NOT supported

Net Numbers are limited to 3 digits (plus the leading 'UN')

There are no net names in this format and each net name is converted into a sequential netnumber with the leading string "UN".

FutureNet Part Types are limited to 16 characters

Designators are limited to 6 characters

Currently only the Pinlist version of the format is supported. Netlist will be added in the future. Both lists contain the same information using a different format.

HiLo Part Names are not checked for length

Net Names are limited to 14 characters

Intergraph No special notes

Mentor Part Names are limited to 19 characters

Module Names are limited to 19 characters

Reference Strings are limited to 19 characters

A separate part file is generated, named <project name>.PRT. Net information is written into a file named <Project name>.NET.

MultiWire Designators plus the Pin Number are limited to 32 characters

Net Names are limited to 16 characters

OrCad PCB II, Part Types are limited to 8 characters

Footprints are limited to 8 characters

Designators are limited to 8 characters

Net Names are limited to 8 characters

Pin Names are limited to 4 charactes

PADS Designators are limited to 6 characters

Net Names are limited to 6 characters

A separate part file is generated, named <project name>.PRT. Net information is written into a file named <Project name>.NET.

PCAD, NLT Net Names are limited 8 characters

Vectron Designators can contain up to 8 characters

Net Names can contain up to 12 characters

A separate part file is generated, named <project name>.PRT. Net information is written into a file named <Project name>.NET.

About netlists
Other netlist formats

Library conversion (Schematic 3)

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.

Organization section of the sheet (OrCAD)

For fields in the Organization section, the following limits will be observed when writing a sheet out to OrCAD format:

The date string is clipped to 18 characters.

The document number is clipped to 36 characters.

The revision string is clipped to 3 characters.

The title is clipped to 44 characters.

The Organization is clipped to 44 characters.

The 4 Address fields are clipped to 44 characters.

(In Advanced Schematic all fields can be 255 characters)

Junctions (OrCAD)

Advanced Schematic supports 4 junction sizes. In OrCAD there is only one junction size so the size information is clipped to the single OrCAD type.

Wires and buses (OrCAD)

There is only one size available for these items in OrCAD, so all width information is lost. Also, line or buses loses their polyline behavior and is broken into individual line segments.

Dashed lines (OrCAD)

Dashed lines are written out to OrCAD as individual Dashed line segments, polyline behavior is lost when saving Dashed lines in OrCAD format.

Bus Entries (OrCAD)

OrCAD bus entries are always 10x10 grid points, so only default size Protel bus entries will be written out as OrCAD bus entries.

Module Ports (OrCAD)

The most important element of OrCAD Module Ports is the length of the Port. In OrCAD the length is totally dependent upon the number of characters in the Port name. In the Advanced Schematic Options-Preferences dialog box, the user can specifiy OrCAD Ports. This simulates the behavior of OrCAD Module Ports so that the port length is dependent upon the number of characters, preserving connectivity when the port is written out to an OrCAD file.

CAUTIONUsers must take care that this option is not enabled when loading sheets created without this option -- because the Ports may be automatically re-sized, changing their connectivity in the sheet. The user can close the file without saving, then reload with OrCAD Ports disabled, to fix this problem.

Inversion of characters in Port names

OrCAD does not support the use of the inverson bar over characters in Module Ports. Advanced Schematic ports with inversion will be displayed as originally entered (backslashes before the inverted character).

Net Labels (OrCAD)

When writing files out in OrCAD format, all font information is lost when the Net name is converted to OrCAD monospace text. Size information is retained using the closest OrCAD font size. In Advanced Schematic, there are 4 orientations for Net Labels: 0, 90, 180, 270 deg. OrCAD restrict Net Labels to horizontal or vertical orientation and characters are not rotated. This can cause distortions when vertical netlabels are written out to OrCAD files. This will not effect netlisting, if the netlabel is placed properly on the wire.

Recommended Practice Restrict net label orientations to 90 or 270 degrees when writing out to OrCAD format.

Sheet Symbols and Sheet Nets (OrCAD)

When writing out sheets in OrCAD format, the same restrictions apply regarding use of the inversion bar in Module Port names (see Module Ports).

Parts (OrCAD)

Most important restrictions involve OrCAD libraries (see Libraries). When writing an OrCAD file, the 8 Description fields, the Sheet Part filename, the Designator and the Part Type. These are clipped to 128 characters in OrCAD (can be 255 characters in Advanced Schematic). OrCAD supports only standard and DeMorgan representations. IEEE mode parts will be written out to OrCAD files as standard mode versions.

Note: OrCAD Power Objects can be loaded as parts (rather than Power Ports) in Advanced Schematic but this is not recommended as it results in inefficient memory use.

Power Objects (OrCAD)

OrCAD supports the first 4 Advanced Schematic types only (Circle, Arrow, Bar or Wave). Power GND, GND Signal and GND Earth are not supported. When saving files in OrCAD format, these will be converted to Bar in OrCAD sheets.

Note OrCAD Power Objects can be loaded in Advanced Schematic sheets as parts (rather than Power Port objects) but this is not recommended as it results in inefficient memory use.

Trace Name (OrCAD)

This primitive is converted into a Simulation Probe when loaded in Advanced Schematic and is converted back to Trace Name when written out as an OrCAD file. The only restriction is on the length of the string (128 characters in OrCAD vs. 255 characters in Protel).

Vector Column (OrCAD)

This primitive is converted into a Simulation Vector when loaded in Advanced Schematic and is converted back to Vector Column when written out as an OrCAD file. The only restriction is on the length of the string (128 characters in OrCAD vs. 255 characters in Protel).

Stimulus (OrCAD)

This primitive is converted into a Simulation Stimulus when loaded in Advanced Schematic and is converted back to Stimulus when written out as an OrCAD file. The only restriction is on the length of the string (128 characters in OrCAD vs. 255 characters in Protel).

Error Markers (OrCAD)

These objects are converted into Protel Error Markers when OrCAD files are loaded and converted back to the OrCAD version transparantly, when written out as an OrCAD file. The only restriction is on the length of the string (128 characters in OrCAD vs. 255 characters in Protel).

No Connect (OrCAD)

These objects are converted into Protel No ERC symbol and converted back to the OrCAD version transparantly, when written out as an OrCAD file. In Advanced Schematic, placing this symbol in Advanced Schematic will supress all error messges, in OrCAD it will only supress Unconnected Pin ERC messages.

Layout Directive (OrCAD)

The OrCAD version of this object is a string only. When loaded from OrCAD the Track, Via and Layer information are split out from the other information. When written back out to OrCAD, these three items are written back into OrCAD format (length is restricted to 128 characters for the total string). Advanced Schematic re-formats Layout information for use with Protel PCB systems and provides additional fields which are not supported by OrCAD.

Libraries (with OrCAD sheet files)

When writing out a sheet file in OrCAD format, the parts list for the sheet is added to the end of the file, as per OrCAD. There is no support in the current version for creating an OrCAD library from Advanced Schematic, so users must create all required components using OrCAD's library management tools.

Loading schematics (OrCAD)

Hierarchy support

Hierarchical OrCAD projects are opened recursively, with each sheet being opened and converted into an Advanced Schematic project. When opening "flat" multi-sheet OrCAD projects, where "Pipestring" links list the other sheets in the project, these are converted into empty Sheet Symbols. Advanced Schematic then "descends" through the Sheet Symbols to open the sheet that was represented by the Pipestring. Therefore, OrCAD flat projects are converted in Advanced Schematic "simple" hierarchical projects, with a Master Sheet and (unconnected) Sheet Symbols for each sheet in the project. The Pipestring is lost. When writing our these projects to OrCAD, they are writting using the same "simple" hierarchical structure. The user must add the Pipestrings back to the master sheet to convert the project back into OrCAD's "flat" format.

Sheet Sizes

In OrCAD, five size labels are available in the system: A, B, C, D, E. These "sizes" do not represent an actual set of dimensions, but can be defined by the user to represent any actual sheet size and stored in the OrCAD environment (not in the sheet file). When loading OrCAD sheets, the user must use Advanced Schematic's custom sheet size option to define the actual required size, as no absolute information is provided by the OrCAD file.

Connectivity (OrCAD files)

OrCAD may not deem some overlapped wires/pins as connected, even when these connections are recognized by Advanced Schematic. All butted connections are recognized in both Advanced Schematic and in OrCAD SDT.

SPICE netlist generation

The SPICE netlist option has some unique options, compared to other netlist formats. These options allow the user to extract the necessary information to generate a SPICE netlist file directly from the schematic.

Net Identifiers

Berkeley Spice currently supports both numeric (Net Numbers) and alphanumeric (Net Names) net identifiers. Earlier versions of SPICE (and some commercial SPICE programs) only support net numbers.

When you generate a SPICE format netlist, Advanced schematic will give the option of net names or net numbers. If you choose to use net numbers, then Advanced schematic will generate net identifiers starting at 10000. If you place net labels on the schematic sheet, they will be ignored unless they are numeric. For example the netlabel 27 will be used in the netlist, but the netlabel VCC1 will be overidden by a numeric net identifier greater than 10000.

If you choose net names instead of numbers then all netlabels from the schematic will be used.

SPICE Text Frames

If you have a text frame with the word "Spice" as the first line, then all the information stored in that text frame will be dumped directly at the beginning of the Spice netlist file.

So, the format is:

Spice

xxxxxxxxxxxxxx

xxxxxxxxxxxxxx

xxxxxxxxxx

etc...

You can have any number of these Spice text frames, anywhere in the schematic project and all will be included at the beginning of the Spice netlist file, so things like model information for components, etc. will be placed here along with directives to the Spice compile, such as probe.

The Spice keyword is not added to the netlist.

OrCAD Pipe directives are converted to text frames when loading OrCAD schematics.

Voltage and Current Sources

In the SPICE.LIB library there are special components for voltage and current sources. These should be used to insert stimulus information into the netlist file. For Example, if an independent

voltage source is required between net 13 and Ground then the required line in the file may be

VIN 13 0 AC 1 SIN(0 0.1 5MEG)

To create this line, set the designator of the volatge source component to VIN, and the part type to "AC 1 SIN(0 0.1 5MEG)" , Advanced Schematic will add the required nodes numbers.

Spice Power Ports

For compatibility with OrCAD, Power Ports can be used to pass information about power inputs to the Spice netlist. If the string "AC" or "DC" appears in a power object, then a special conversion is done on the string net power object and it is written to the Spice netlist in a special format. For example, if you have a power object and the string in the power object is "VCC DC 12" then one of the nets that is created as VCC, will have the attribute DC 12 attached to it in Spice format. In addition, in the MAP file, that same information will be written out as the net

name VCC DC 12.

Map files

The Spice format creates two MAP files in addition to the netlist files. A list of the node numbers, cross referenced to the net names is placed in the first MAP file because the netnames

in the Spice file are numbers starting at 10000 and incrementing sequentially, because many Spice systems can't handle alphanumeric net names. Also, it's important to use the Spice library,

because Spice systems can only support numeric node numbers (or pin names). So, for example, if you have a transistor, you have to convert E,B and C to 1,2 and 3, etc. Protel will add a library

called SPICE which will have components in those formats. Users should either restrict component use to the Spice library, or modify their transistor libraries, etc.to support those conventions.

About netlists
Other netlist formats
Netlist format notes

See also
Create Netlist

Choose Schematic Sheet

New Features (Release 2.0)

A number of new features have been added to Advanced Schematic. Select a topic for information about these features:

Concepts

Additional Part Fields
Additional Sheet Options
Additional Special Strings
Additional System Information
Advanced Pld Netlist
Advanced Global Editing For Parts
Automatic Junctions
Autopan Options
Command Status Bar
Communication between Applications
Complex Hierarchy support
Component Browser Changes
Cross Probing
Cross Project Global Editing
Direct Selection
Drag and Drop editing
EESOF Support
Environment Files
File Dialog Changes
File Open Filters
Font Manager
Forward Annotation to Advanced PCB
Graphical Editing Hotkey List
Guided Wiring
Improved Connection Dragging
Memory Monitor
Object Alignment
OrCAD SDT 386+ support
Part Field Names
Part Update reporting
Project Manager Changes
Protel 2 Netlist changes and additions
Quick copy attributes
Re-entrant editing commands
Save Object Defaults
Sheet Templates
Special links to PCB systems
SPICE changes and additions
Tools Menu

Commands

Using Protel Advanced Schematic With HP EEsof Touchstone and Libra

Advanced Schematic includes netlist output format for HP EEsof Libra and Touchstone and a simulation model library, EESOF.LIB. Advanced Schematic outputs two EEsof formats: EEsof Touchstone, for linear simulation; and EEsof Libra, for non-linear simulation. Both EEsof netlist formats are saved using the special file extension ".CKT."

Setting component parameters

Component objects have been specially extended to encompass the EEsof simulation parameters. Users can define simulation parameters in an existing part using the extended part field controls in the library editor and the Part dialog box.

Attributes fields region of a Part dialog box for an EEsof part. In this library, part fields have been named for the simulation parameters. These parameters are compiled into the CKT block of an EEsof Libra or EEsof Touchstone netlist.

When an EEsof netlist is output from Advanced Schematic, any text in the part fields of a component are compiled directly into the netlist. These parameters are defined in the dialog box for the part in the part fields and using the part field names definable in the library editor. See the "Part Fields" section for more information.

When an EEsof netlist is output any text in the part fields of a component are compiled into the CKT section of the netlist and an equal sign "=" is automatically placed between the part field name and the part field text.

Several conventions are used to conform to the syntax of the EEsof netlist. See your EEsof documentation for more information on netlist syntax, usage and parameter definitions. The following marks are used in part field names (set from the library editor) to indicate the use or syntax:

{} indicates an optional parameter; the Part Field Name is not added to the CKT section if a value is not specified.

< > indicates the text in the part field will appear in the CKT section without the Part Field Name, used when specifying datafiles.

[ ] indicates the Part Field Name and the text in the part field will be enclosed within open and closed brackets.

(=) indicates no "=" (equal sign) will be inserted between the part field name and the text in the part field.

The following mark is used only in a part field:

^ inserted before a character in a part field overrides the default "=" inserted between the part field name and the part field.

Source Block Components

Source Components that are to be included into the Source Block of the CKT file need to have "(Source Block)" included in the description field for the part.   This is defined in the Library Editor.

Embedded text blocks

You can embed text in a schematic which will be compiled into an EEsof netlist by placing a text frame(s) on any part of your project. Each text frame must start with the syntax "EESOF_<DataBlockName." on the first line (by itself) and any further information placed on subsequent lines, per the example below.

Embedded text frames are used to place information into a section of an EEsof netlist, specified by the data block name, on the first line of the text frame. This example is from the file EESOF3.SCH.

General simulation settings

When the EEsof Touchstone or Libra netlist format is designated during netlisting, a special dialog box is displayed:

A dialog box opens when the File-Create Netlist   command is used and an EESof format is designated. This dialog box allows users to select the desired unit values for each simulation measurement field. For example, for Frequency the designer can choose from the units GHz, Hz, KHz or MHz. This information is attached to the DIM section of an EEsof netlist.

See also

Create Netlist

Automatic Junction Insertion and Removal

When enabled, automatically places a junction tie when placing wires or other electrical items. Junctions are placed when a wire is placed across another object's electrical hot spot(s) (e.g. across component pins, power ports, etc.). Junctions are also placed when a wire is terminated on another wire segment. When this option is active, junctions are also removed automatically when electrical object connections are broken.

See also

Options Preferences

Sheet Templates

Sheet templates are created in the same environment used for schematic editing. Once created, these pre-defined templates can be applied to new or existing projects. To define a template, first choose File-New to load an empty sheet into the workspace, then:

1. Choose Options-Sheet from the menu bar. Unmark the Title Block check box. Click OK.

Notice that the standard title block no longer appears on the page. Zoom-in to the bottom right corner of the page to start a custom title block. Draw a new title block:

2. Choose the graphical line tool from the Drawing Tools palette or use the Place-Drawing tools line command.

3. Before starting the line, press the TAB key. The Line dialog box opens.

4. Click in the Color box to open the Color Selector, then scroll up to color number 4 (black). Click OK to close the Color Selector dialog box then click OK in the Line dialog box to accept this change.

5. Now, position the cursor in the sheet workspace and click to begin the first title block line segment. Click to mark the end of each segment, continue until you have defined a box.

6. Press ESC key once to end the line and press ESC again to exit the Place Line command. Now place text in the title block:

7. Choose Place-Text-Annotation from the menu bar.

8. Before placing the text, press the TAB key to change the text attributes. Press the Font Change button.

9. In the Size field, type 10 and Click OK.

10. In the Text field type .ORGANIZATION and then click OK.

The special string .ORGANIZATION is mapped to a field in the Document Info dialog box. This box is accessed from the Options-Sheet dialog box. Use of special strings is outlined in the next tutorial, below. Continue to define your custom title block:

11. Position the cursor inside your new title block, then click OK. Press ESC to exit the place text annotation command. Now save this sheet as a template:

12. Choose File-Save As from the menu bar. Type MY_SHEET.DOT in the Filename field and click OK.

The extension .DOT defines this file as a sheet template. This template can be loaded into a single existing sheet or an entire hierarchical project. Open a new sheet and load your template:

13. Choose File-New from the menu bar. A new schematic sheet is created. Now load the file MY_SHEET.DOT as a template.

14. Choose Options-Load Template To Sheet from the menu bar. Choose MY_SHEET.DOT and click OK.

15. Choose Options-Sheet from the menu bar. The Schematic Sheet dialog box appears.

16. Mark the Enable Sheet Template check box and click OK.

This new sheet now include the template information you saved. Sheet templates can be created from any schematic sheet at any time.

See also

Set Template File Name
Update Template
Remove Template
Options Preferences

Cross Probing

Advanced Schematic fully supports multi-document and multi-application use of the Windows environment. This allows the user to open Advanced Schematic and Advanced PCB at the same time. When a schematic sheet (or project) and its PCB layout are open at the same time, Advanced Schematic and Advanced PCB version 2 support bi-directional cross probing.

For example, the user can select a part in the schematic, and the PCB editor will jump to and display the corresponding component. The corresponding PCB-to-Advanced Schematic cross probe is also supported: pick a component footprint on the PCB and display the corresponding schematic part. Schematic pin-to-PCB pad cross probing and net label-to-physical net cross probing are also supported.

See also

Communication with Advanced PCB
Cross Probe Part
Cross Probe Pin
Cross Probe Net

Find and replace text editing

Advanced Schematic allows users to find and replace text anywhere on a sheet or across a multi-sheet project. For example, you may wish to rename a net across a multi-sheet project.

To search and replace text:

1. Choose the Edit-Replace Text command. The Text Find And Replace dialog box opens.

Text Find and Text Find And Replace dialog boxes are similar. This process supports the use of wildcards (*) when searching for text and allows the use of conditional replacement x=Y.

2. Type the Text To Find string.

The Find Text and Replace Text commands support the use of the asterisk "*" as a wildcard character for variable length text strings (see below).

3. In the New Text field, type the replacement text. Replacement can be made conditional (see below).

4. 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, if desired.

5. Choose any other Options. Changes can be made on a Case Sensitive basis (upper and lower case must match exactly when searching). Replacement text always matches the case used whentypeing text into the New Text field. If you wish, you can have Advanced Schematic prompt you before the replacement is made for each item.

Restricting changes to net identifiers

This option allows the user to restrict find and replace text editing to net identifier objects: net labels, power ports, ports and sheet entries.

6. Click OK to perform the text replacement.

See also

Find Text
Replace Text
Find Next

Component Browser Changes for version 2

The Browser panel is provided as an aid in searching for, and placing component parts into sheets and for locating other objects during schematic editing. This panel can be displayed as needed or hidden, to free more of the display area for the sheet workspace. The Browser has three sections, Library, Components In Library and Browse.

Additions to Components In Library

Find Button - Opens the Find Component dialog box, this command is used to search for a component that is in a specified drive/path, in the current listed libraries, or on all drives across a network. The component can be found by library reference name and/or by component description.

Additions to Browse

All electrical objects can now be listed in the Browse list box.

Whole Project checkbox - specifies if all objects in a project are listed or only the objects on the current sheet.

Mask field - used to list specific objects. Wildcard characters (*) can be used in the Mask field.

Partial Info checkbox - if enabled, only partial object information is listed in the object list box.

Text button - Click the text button at the extreme bottom left of the Browser panel to change the text of the designator (for parts) or label (other objects with text). If the object does not have a text label, clicking the button is ignored.

Jump button - Click the Jump button to jump the the highlighted primitive name in the list box. The display will pan to center the object and the object will be placed in focus, displaying its graphical editing handles or control points.

Edit button

Click the Edit button to open the primitive's dialog box, which provides access to all object attributes.

See also

Edit Object From Browser
Find Component in Library

Guided Wiring

Special automation features speed the connection of electrical items in the schematic sheets. An electrical grid provides true "snap to" wiring of all electrical items: ports, sheet entries, buses, bus entries, net identifiers, wires and parts. When this feature is active, the cursor will jumps to any nearby electrical "hot spots" and then change shape to indicate the connection point. The user need only click (or release LEFT MOUSE) to complete the connection.

Options allow the user to turn the electrical grid off or on and to automatically place junctions when terminating a wire tangent to an existing wire. Junctions are also automatically placed when wires cross pins of parts or power objects. Connections between electrical objects are maintained as the objects are "dragged" to a new location on the sheet. The system will even add or remove wire segments to maintain orthogonal routing during complex moves.

See also

Options Sheet
Options Preferences

Spice Netlist Changes and additions for verison 2

See also

Create Netlist

Part Field Names

The names of part text fields can be defined in the Schematic Library Editor (Component-Description command). Up to 255 characters can be used, but only the end of the character string will be displayed in the Part dialog box if the length of the string exceeds the display area. There is room for 14 characters and/or spaces, when 12 point 'Helvetica' is used as the dialog box font (Options-Preferences command). Custom field names are not used for column headers in CSV format Bill of Material reports.

See also

Part Fields

Protel-2 Netlist Format

This option is an extended version of the Protel netlist. It is distinguished by the additional field support and detail that supports Advanced PCB 2.0 and simulation packages. Note that this format has three sections and that each field is first named, with the field data on the following line.

The first part of a Protel netlist describes each component

PROTEL NETLIST 2.0 File header identifies Protel 2 format;

[Bracket marks the start of each component

DESIGNATOR Each field is first identified;

C8 Component designator (label);

FOOTPRINT

RAD0.2 Footprint/package description

PARTTYPE

0.1uf Comment, (or value);

DESCRIPTION

* Sheet part description field;

Part Field 1

* Sheet text fields (1-16) of up to 255 characters;

Part Field 2

*

(etc.)... (continues to Part Field 16)

Part Field 16

*

LIBRARYFIELD1 Library text fields (1-8) of up to 255 characters;

LIBRARYFIELD2

(etc.)...(continues to Library Field 8)

LIBRARYFIELD8

] End component delimiter;

The component description section is followed by a listing for each net within the netlist.

(Begin net delimiter;

H/-E Name of first net;

DECA1-1C DEC36-1C PASSIVE First node in net. Includes: component-pin designator (single blank space) part type-pin name (single blank space) Pin electrical type;

U16-1 74HC00-_A INPUT Next node in net;

U16-2 74HC00-_A INPUT Last node in net;

) End net delimiter;

Any number of components or nodes can be included in a Protel or Protel2 netlist, limited only by available memory.

Note that net descriptions are distinguished from component descriptions by the use of rounded, rather than square brackets. The extension .NET is reserved for Protel format netlists. A simple **Protel netlist format**, supports basic netlist-based auto component placement and autorouting in Protel Autotrax and Protel for Windows PCB layout packages.

See also

Create Netlist

Advanced Pld Netlist

Protel's Advanced PLD application uses this netlist to compile logic for FPGA/PLD programming. This format also supports the CUPL FPGA/PLD environment from Logical Devices, Inc. The format is the input to the LIAISON program from Logical Devices, Inc.

See also

Create Netlist

Object Alignment

Two alignment methhods are provided. You can align a group of selected objects on both axis, by choosing Edit-Align-Align. Or, you can align objects on one axis, by choosing the other Align menu commands.

Align objects by selecting them and then choosing Edit-Align command, or by choosing Edit-Align--Align Left, Align-Right, etc.

To align 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 Edit-Align-Align and designate 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.

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

To align objects using the other Align commands:

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 Edit-Align and the desired alignment command (Align Left, Align Right, etc).

3. The selection will be aligned according to the command selected.

Remember, if you are not happy with the alignment, you can use Edit-Undo to remove this change.

See also

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

Popup Alignment Menu

Part Fields Changes for version 2

Parts have special text fields, some of which are created and stored at the library level and others that are created and stored at the sheet level when the part is placed. Advanced Schematic supports sixteen part (sheet-level) text fields and eight component (library-level) text fields, of up to 255 characters each.

Special text fields, which are included in netlists: designator (label) type (description) and package (footprint), may be limited in the number of characters supported. These netlist fields may or may not support empty spaces and/or be case sensitive.

When parts are placed in sheets the designator, type description and other text fields created at placement can be either hidden, displayed or moved relative the part position and orientation. Displayed text supports any TrueType font, including user-definable size and style.

Library text fields

Each library component has eight user-definable text fields. These fields can hold up to 255 characters each. Library text fields cannot be edited from placed parts in schematic sheet files, but are automatically included in CSV format Bill of Materials reports. An additional 16 text fields are editable for each instance of a part, placed in sheets. These part fields can be viewed (along with other read only attributes) from the Parts dialog box after placement in schematic sheets by clicking the View Read-Only Library Fields button.

Part text fields

In addition to the eight library component text fields, sixteen additional user-definable fields are available for each part, at the sheet level. These fields can be displayed or hidden (clickHidden 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 automatically included in CSV format Bill of Materials reports.

See also

Part Field Names

Communication with Advanced PCB

Advanced Schematic supports full forward annotation of PCB files in Advanced PCB. Updated netlists can be used to make engineering changes to fully or partially routed PCBs. See your Advanced PCB 2.0 documentation for more information.

The Protel 2 netlist format is supported by Advanced PCB version 2.0 or later and includes routing directives defined in the schematic.

Bi-directional cross probing

Advanced Schematic supports cross probing of Nets, Pins and Parts to and from Advanced PCB 2.0. You can use Schematic/PCB cross probing while running Advanced Schematic and Advanced PCB simultaneously, with a PCB file open and its source schematic open. Cross probe from either program by choosing Edit-Cross Probe, and clicking on an object. Missing components or pins (PCB layout) Advanced PCB reports missing component patterns or missing pins when two factors are present in the netlist:

One or more Package Descriptions (or Type) is missing from schematic component information in the netlist, or the package in the schematic does not match any Advanced PCB library component. The names of missing components and pins will be listed in a netlist report file (an option, when you load a netlist in Advanced PCB).

It may be necessary to re-edit the schematic or netlist to include the Type information, or additional Advanced PCB library components may be created to match any unique descriptions in the netlist.

If all components are present but pins are reported missing, the cause is usually that the schematic package's pin numbering differs from Advanced PCB.

Schematic libraries contain specific components and devices. PCB footprint (decal) libraries contain generic footprints which can belong to various specific components -- each having different pin assignments.

For example, a transistor shape can represent various combinations of "E," "B" and "C" -- each of which must be assigned to the correct pin number in the PCB layout system. Capacitors are a similar case, with pins often named "A" and "K" in the schematic. One solution is to leave the schematic pin designations as "E," "B" and "C" and then place components on the PCB and change the pad designators to match. If you have a lot of pin outs in the same orientation, you may want to make a special version of the component in the library using the correct pin identifiers.

See also

Cross Probe Part
Cross Probe Pin
Cross Probe Net

Drag and Drop editing

Advanced Schematic allows quick "drag and drop" editing of placed objects. Any placed object can be moved with a simple point, click and drag sequence. Test this feature on any placed item:

1. Move the cursor over the object.

2. Press and hold LEFT MOUSE, then drag the object without releasing the LEFT MOUSE button.

3. Release LEFT MOUSE. The drag object feature allows quick movement of schematic objects but does not maintain connectivity while moving.

When moving a connected electrical object, maintain connectivity with the drag object feature by holding down CTRL while clicking LEFT MOUSE. Once the object begins to move, you can release the CTRL key.

Quick copy attributes

The "quick copy attributes" feature allows you to copy the attributes of one object into a second object of the same type. To demonstrate this feature place a port on a sheet:

1. Double click on the Port to open its Port dialog box.

2. In the Name field, type Port_1.

3. In the Width field, type 90.

4. Click OK to accept these changes and close the dialog box. Now place a second port on the sheet.

5. Chose Place-Port or click on the Port button on the Wiring Tools palette.

6. Move the new port over the first port, so that the center of the cross hairs is within the boundaries of "Port_1."

7. While holding the port over "Port_1," press the INSERT key. The new port will jump "over" the first port.

8. Move the new port away from the original, then finish placing the port by clicking to define its length. All attributes of "Port_1" should now be copied into the new port. If the port has not "inherited" the attributes of the first port, the cursor crosshairs may not have been inside the first port when pressing INSERT.

The quick attribute copy function can be used to "clone" the attributes of all placed schematic object types, except sheet symbols. This process works whether placing a new object or can be used when moving an already-placed item. It does not work when dragging connected electrical objects.

Connection Dragging

Connectivity between electrical hot spots is automatically maintained. To examine this feature:

1. Place any part on a schematic sheet. The type of part does not matter.

2. Place another part so that any pin on the first part butts up against a pin on the second part. Press ENTER to place the second part and then press ESC to cancel the Place Part command. When any two hotspots are connected, as in this example, the connection will be maintained when dragging any connected electrical item. To see this feature in operation, drag one of the components away from the other:

3. Hold down CTRL and click LEFT MOUSE over one of the parts. Release the CTRL key and the mouse button. The system is now in Drag Single Item mode.

4. Now, move the mouse slowly to drag one part away from the other. At the moment the part is moved, the point where the two pins were connected was transformed into a wire, which stretches as you drag to maintain the connection. As you move the part, special rules govern the placement of the wire. These rules are applied similarly to using the Place-Wire command. You can change the placement rules while dragging by pressing the SPACEBAR.

5. While dragging the part, press SPACEBAR to toggle between the four available wire placement modes.

Modes include Any Angle, 90 degrees, 90 degrees constrained on one axis or 90/45. Connection maintenance is an invaluable tool for quickly connecting any two electrical objects: parts, ports, wires, power objects, busses, bus entries, and sheet symbol entries.

See also

Dragging Selections
Dragging Objects

Re-entrant editing commands

Advanced Schematic allows users to execute commands from within other commands. A simple application of this feature is to change views while placing schematic objects. In continuing this example, you can test this feature by placing a sheet symbol while dragging connections. Begin by again dragging the left most part to the upper right-hand corner of the screen.

1. Choose Edit-Move-Drag from the menu bar and drag the part on the left to the top right corner of the screen.

Now place an new schematic object while still dragging the part. Use the keystroke shortcuts to place a sheet symbol:

2. Press P and then the S key to start the place sheet symbol command.

3. Press the TAB key to change the attributes of the sheet symbol before you place it. The Sheet Symbol dialog box appears.

4. In the Filename field, type SCH_3.SCH.

5. In the Name field, type SubSheet1.

6. Click OK once to continue placing the sheet symbol.

7. Place the sheet symbol in the bottom left corner of your screen. Make the sheet symbol box 0.5-inch by 0.5-inch (5x5 grid units on a 0.1-inch grid) by moving the cursor, clicking the LEFT MOUSE button once to set the height and moving and clicking a second time to set the width.

8. Press ESC once to exit the Place Sheet Symbol command.

Now finish the drag connection function by dragging the part under the cursor to the upper right corner.

9. Finish the drag connection command by pressing ENTER.

Re-entrant editing commands lets users to work more flexibly and intuitively. For instance, a user can start placing a wire, then remember that it is to be connected to a port, place the port and then connect the wire to it.

Any number of commands may be executed within another command and up to three uncompleted commands can be nested within each other.

Direct Selection

Hold down LEFT MOUSE and drag a selection rectangle over the desired area in the sheet. All objects inside the selection rectangle will be added to the current selection. To delete objects from the selection, hold LEFT MOUSE+SHIFT while dragging a selection rectangle.

Remember, these processes can be re-assigned to different mouse/key shortcuts in the Options-Hot Keys dialog box. You can use the Zoom commands (and hot key shortcuts PGUP and PGDN) at any time when making or releasing selections.

Special Strings

Special text string are used to place automatically updated information or information that can change from sheet to sheet. Support for special strings is provided for use in sheet templates. The MY_SHEET template created previously, included the text string .ORGANIZATION in the title block. This special string is linked to a text field that can be accessed from the Options-Sheet command. With the new sheet open and the MY_SHEET.DOT template loaded, enter your company information for this sheet:

1. Choose Options-Sheet from the menu bar.

2. Click once on the Document Options button.

This dialog box holds information that can be displayed on a schematic sheet automatically with the special text strings. Enter your company information in the Organization field:

3. In the Organization field, type the name of your company. Click OK. The view returns to the Schematic Sheet dialog box.

4. Click OK to accept the change and close the dialog box.

Now convert the strings so that they display your company information:

5. Choose Options-Preferences from the menu bar.

6. Mark the Covert Special Strings check box and click OK.

When you return to the sheet, note that the string .ORGANIZATION has been converted to the company name you typed in the Organization dialog box, above. Special text strings can be placed in sheet templates or on normal sheets.

The following special strings automatically display the information entered in the Document Options dialog box, accessed from within the Schematic Sheet options dialog box:

.ORGANIZATION Lists Organization field text.

.ADDRESS1 Lists text from first Address field.

.ADDRESS2 Lists text from second Address field.

.ADDRESS3 Lists text from third Address field.

.ADDRESS4 Lists text from fourth Address field.

.SHEETNUMBER Lists text from Sheet No. field.

.SHEETTOTAL Lists text from Sheet Total field.

.TITLE Lists the Document Title text.

.DOCUMENTNUMBER Lists the Document No. text.

.REVISION List the Document Revision text.

The following special strings automatically insert the current information at the time the document is printed.

.DOC_FILE_NAME   The name of the schematic sheet file.

.TIME   The current time.

.DATE   The current date.

See also

Options Sheet
Options Preferences

Sheet Options

The Schematic Sheet dialog box is used to assign sheet-level preferences and defaults (these are items associated with a single sheet, rather than the system level preferences described above).

The Options Sheet command opens the Schematic Sheet dialog box which is used to define sheet level defaults. These assignments are saved with the sheet and are also used as defaults when a new file is created.

Style

Selects from available standard sheet formats.

Use Custom Style

Uses Custom Width, Height, Ref (Reference) Region count and Margin Width settings. Units are .01 inch. Ref (Reference) Region count refers to the number of divisions in sheet borders. Border references are indicated by 1, 2, 3, etc. (X axis) and A, B, C, etc. (Y axis). References begin at the sheet origin (lower left corner) Margin refers to the area between the border reference and the edge of the sheet.

Colors

Colors can be independently assigned to the sheet background and to the template items (border and title block regions). Clicking in the color box opens the Color selector dialog box. Any of 224 pre-defined or user-defined RGB colors can be assigned.

Grids

Any sheet can have a pre-defined Snap grid and Visible grid pitch.

Electrical Grid

The special electrical Snap grid allows the cursor to jump to the nearest electrical hot spot when placing or moving electrical objects (wires, buses, ports, parts, etc.). The Grid Range value determines how near the cursor need be to snap to the nearest electrical hot spot (units of .01 inch). Other sheet options

Orientation

Designates either Landscape (default) orientation or Portrait orientation. Landscape is "wide" orientation, Portrait is "tall" orientation.

Title Block

Designates either Standard or ANSI format for the title block region. Users can define custom title blocks and save these using template (.DOT) formats, which can be applied to any new or existing sheets.

Show Reference Zones

When enabled, this option includes the reference information (1, 2, 3, A, B, C, etc.) divisions within standard or custom borders. Has no effect when the border region is disabled (see below).

Show Border

When enabled, the border region is displayed.

Show Template Graphics

When enabled, displays any linked graphics associated with the current template. Disabling this option can speed sheet re-draw, if the template includes large or complex graphics.

Document Info...

Opens the Document dialog box. This dialog box provides access to all organization information. This information is used to pre-define the contents of sheet title blocks.

Change System Font...

Opens the Font Manager. Allows user to define a font, font size and font style for all system-level sheet text, including port names, power port names, pin numbers and sheet entry names.

See also

Options Sheet dialog box

Forward Annotation to Advanced PCB

Forward annotation is the process of implementing changes to an existing PCB layout from the schematic editor. This system will allow users to move, add or delete connections on the schematic sheet. Adding, removing or re-naming of parts or nets in the schematic is also supported. When the updated netlist is loaded into Advanced PCB 2, the system will compare the new netlist with the physical design and update the PCB to reflect all netlist level changes. This can include substitution of component footprints and removal of obsolete routed tracks with conversion back into unrouted (ratsnested) PCB connections.

Special links to PCB systems

Advanced Schematic supports a number of links to PCB systems beyond basic netlisting.

Back annotation

The File-Back Annotate command updates schematic part designators (U1:1, R32, etc.) based on a "was-is" list generated by the PCB layout package. Advanced PCB 2 and a number of other layout packages support back annotation. The "was-is" information is supplied by a .WAS file. This file is in ASCII text format.

Forward annotation

Forward annotation is the process of implementing changes to an existing PCB layout from the schematic editor. This system will allow users to move, add or delete connections on the schematic sheet. Adding, removing or re-naming of parts or nets in the schematic is also supported. When the updated netlist is loaded into Advanced PCB 2, the system will compare the new netlist with the physical design and update the PCB to reflect all netlist level changes. This can include substitution of component footprints and removal of obsolete routed tracks with conversion back into unrouted (ratsnested) PCB connections.

PCB Layout directive

The PCB Layout object (supported by OrCAD SDT and PCB) allows the engineer to specify topology, priority and design rules for routing on a net-by-net basis. User definable fields for the PCB Layout object include routing track width and via size, layer and route priority. Directives are included in the Protel 2 netlist format. Advanced PCB (version 2) is fully-compatible with the Protel 2 netlist format.

Cross probing

Advanced Schematic fully supports multi-document and multi-application use of the Windows environment. This allows the user to open Advanced Schematic and Advanced PCB at the same time. When a schematic sheet (or project) and its PCB layout are open at the same time, Advanced Schematic and Advanced PCB version 2 support bi-directional cross probing. For example, the user can select a part in the schematic, and the PCB editor will jump to and display the corresponding component. The corresponding PCB-to-Advanced Schematic cross probe is also supported: pick a component footprint on the PCB and display the corresponding schematic part. Schematic pin-to-PCB pad cross probing and net label-to-physical net cross probing are also supported.

See also

- **Cross Probe Part**
- **Cross Probe Pin**
- **Cross Probe Net**
- **Create Netlist**
- **PCB Layout Directives**
- **Back Annotate**

Windows .WMF Clipboard support

You can use the Windows clipboard to Cut, Copy or Paste sheet elements, text or graphics from one sheet to another. Windows metafile (.WMF) support allows you to use the clipboard to Paste selections from Advanced Schematic sheets into other applications. You can also copy text from other applications, via the Windows clipboard and paste this text into Advanced Schematic text frames. An Add Templates to Clipboard option (Options-Preferences command) allows the user to Copy an entire sheet (including border and title lock) to the clipboard.

See also

Edit Cut
Edit Copy
Options Preferences

Cross Project Global Editing

Advanced Schematic supports global editing of schematic objects throughout a multi-sheet, hierarchical project. This feature allows users to change items located in various parts of a project or impose style changes throughout all open sheets in the current project.

Change Scope

These three options define the extent of changes:

Change This Item Only (no global changes);

Change Matching Items In Current Document;

Change Matching Items In All (currently opened) Documents.

The latter option extends your changes across all open sheets, allowing you to globally edit an entire project.

Complex Hierarchy support

Advanced Schematic supports single sheet, multiple sheet and fully hierarchical designs including complex hierarchy where multiple instances of a single sheet file can be used in a project.

Multi-sheet projects can be opened, closed or saved with a ingle File menu command. Projects can be navigated visually using the Project Manager, a special display window that resembles the Windows File Manager. The Project Manager displays all open files in a hierarchical tree structure. Users can click on sheet icons to move from sheet-to-sheet.

The Project Manager displays three types of sheet icons:

Master sheets - These icons represent the top level of hierarchical projects or "flat" multi-sheet projects, which have two levels only.

Subsheets - These icons represent sheets that are "descend" from master sheets.

Complex sheets These icons represent duplicates of other subsheets, used in complex hierarchies. When you click on a Complex sheet icon, the focus moves to the subsheet it represents. Colors can be assigned to these three icon types (Options-Preferences command).

See also

Options Preferences

Trace Netlist Generation Options

The Trace Netlist Generation option is a special design validation tool that allows users to examine details of netlist generation. This allows the user to track netlist discrepancies back to a specific cause or condition.

The Trace Netlist Generation options are in the Create Netlist dialog box.

Enable Trace - turns netlist tracing report on or off. If enabled, the trace information is written to <filename.tng>

Netlist before any resolving - internal netlist after spatial to connective conversion, but before any merging of nets.

Netlist after resolving sheets - internal netlist after merging only inside sheets, but before any merging of sheets into one project.

Netlist after resolving project - internal netlist after merging and resolving of nets inside a project.

Include Net Merging Information - if enabled all merging information is written to the report file.

See also

Create Netlist

Project Manager Changes for version 2

You can click on any sheet icon to move the current focus to that sheet window. You can also use the (up, down, left, right, home, end, pgup, and pgdn) keyboard keys to move up and down the project hierarchy tree.

The Rebuild button at the top of the Project Mananger window, is used to update the hierarchy tree when changes are made to the projects organization (as when adding or deleting sheet symbols from parent sheets).

Expanding and Collapsing the Project Hierarchy Tree

In the Project Manager, you can view the subsheets by expanding the hierarchy tree. You can also choose not to view the subsheets by collapsing the hierarchy.

To display subsheets.

Double-click the parent sheet you want to expand. The sheets that have subsheets are marked with a plus sign (+).

Or select the sheet you want to expand and then choose the Expand One Level (->),   Expand Branch (-->>), or Expand All (All) button at the top of the Project Manager.

To hide subsheets

Double-click the parent sheet you want to collapse.

Or select the sheet you want to collapse and then choose the Collapse Branch (<<--) button at the top of the Project Manager.

The width of the Project Manager can be changed by dragging its right hand side toward the center of the screen.   The cursor will change into a left-right arrow when the cursor is over the right edge of the window. To see more of the hierarchy tree, hold LEFT MOUSE and drag to adjust the width.

Project Manager display options

The Project Manager displays three types of sheet icons:

Master sheets - These icons represent the top level of hierarchical projects or flat multi-sheet projects, which have two levels only.

Subsheets - These icons represent sheets that are descend from master sheets.

Complex sheets These icons represent duplicates of other subsheets, used in complex hierarchies. When you click on a Complex sheet icon, the focus moves to the subsheet it represents. Colors can be assigned to these three icon types (Options-Preferences command).

See also

Options Preferences
Toggle Project Manager

File Open Filters

Clicking the button under File Types allows you to assign a new a new file mask. For example, you may wish to change the defaultextension to search for other file types. When adding a new file type the description can be any character and the extension must be enclosed within round brackets.

File Dialog Changes

The File dialog boxes feature a number of control options, including direct access to the Windows File Manager (for browsing and modifying directories), and networks (for accessing networks and network drives). Clicking the button under 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.

Information about the specified file (the filename currently highlighted by the selection bar in the File list), is displayed in the bottom of this dialog box. 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.

Font Manager

Using fonts

Advanced Schematic directly supports the TrueType fonts that are delivered with Windows, including bold and italic formats and display font 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.

Font Style dialog box

Advanced Schematic includes a powerful Font manager utility which allows users to monitor the available (installed) windows fonts and make intelligent font selections for both display and printing of schematic sheet. Font manager routines are available whenever you access the font assignment options for sheets or for design objects that include user-definable text. These objects include text annotation, text blocks, components, etc. Clicking the Font Change button (in the object's dialog box) opens the Font Style dialog box. This dialog box displays installed system fonts (by font type) including the fonts available for the current output device (if the driver for the device includes font information). The user can examine the installed fonts, change the font size and style or change the current output device from this dialog box. If you have problems rendering fonts used in the sample sheet files supplied with Advanced Schematic, make sure that the appropriate TrueType fonts have been installed and enabled. If a font is not rendering properly, double click the object and click Font to see the name of the font. You can assign another available font or check to see it the font is enabled.

Fonts are installed from the Windows Control Panel, Fonts utility. To enable installed TrueType fonts, click the TrueType button and then the Enable TrueType fonts checkbox in the TrueType dialog box. See your Microsoft Windows Users Guide for details about font management under Windows.

Font Management System

Font management in Advanced Schematic gives users extensive information about available font resources and capabilities. Users can exercise broad control over font usage, including the ability

to apply a System font used for rendering part pin names, port names, power objects and sheet (border) references.

Font Technology

Font technology and behavior can vary widely within the Window environment. Two main types of fonts are used in the Windows environment: Screen fonts, which are used by a display driver to show letters on a monitor, and printer fonts, which are geometric

descriptions of letters used by a printer to draw text on a page. Since screen fonts and printer fonts are designed for specific purpose, they can produce undesirable results when used with a device for which they were not intended. For instance, if a font intended for screen use only is used on an high-resolution printer, the text will have jagged, blocky letters.

Font Manager

The Font Manager gives users more control over font resources and provides extensive information about font technologies for more predictable display and output results. Advanced Schematic's font management system allows users to access printer-resident fonts, even if no screen font is available. If you choose a printer-resident font that lacks a screen representation, the font will be displayed on screen with the closest matching screen font.

Advanced Schematic's font management system recognizes and identifies TrueType, Vector fonts (including PostScript and printer/plotter-resident fonts) and Raster font technology.

TrueType fonts describe letters using vector outline descriptions of letter shapes and are managed directly by Windows. These fonts scale smoothly, can be rotated include a display version for accurate on-screen rendering (what you see is what you get).

Vector or plotter fonts used vector rather than bitmap descriptions. Vector fonts are used primarily by plotting devices and some dot-matrix printers. These fonts are scaleable and rotatable, but usually do not include accompanying screen font.

Raster or bitmap fonts describe letters with small dots on printer or pixels on a monitor. Raster fonts are primarily intended for screen display and cannot be rotated.

PostScript fonts are similar to TrueType fonts. These outline fonts are not directly supported by Windows and must be managed using a system extension such as Adobe Type Manager for Windows. PostScript outline fonts are downloadable to PostScript compatible printers allowing you to print smooth representations of letters even if the font is not resident in your printer. These fonts scale smoothly, are rotatable and are usually accompanied by a screen font for accurate screen rendering.

System font control

With Advanced Schematic, users can set the default font for sheets. The Options-Sheet dialog box has been expanded to include a Change System Font control, which allow users to control the standard font used for boarder display and title blocks. The system font controls use the same font dialogs used in the new font selector dialog.

See also

Options Sheet

Tools Menu

The Tools menu allow users to run other Windows applications and utilities from the Advanced Schematic editors. The Tools menu can be configured by choosing Tools-Setup from the menu bar.

Configure the Tools menu by typing the name of the executable for an application file in one of the fields of the Setup Run Options dialog box. If the program you wish to run is not in the DOS path, you must type in the complete path (limited to 255 characters).

See also

Setup Run Options

Part Update reporting

Updates of parts from schematic libraries are recorded in a report file. When the command Library-Update Parts In Cache is executed, a report file is generated which lists parts that were updated. This list is automatically saved and displayed in Windows Notepad. The report only includes components that where actually changed.

See also

Update Parts from Library command

OrCAD STD 386+ support

Orcad 386+ schematic sheet files and library files must be converted to Orcad 16bit files before they can be loaded into Advanced Schematic.

When importing Orcad 386+ schematic files, Advanced Schematic looks in the current path for Orcad's 32TO16.EXE utility, if found, Advanced Schematic uses this utility to automatically convert and load the 32bit schematic file. The 32TO16.EXE utility also requires the Orcad file SDTXTEND.EXE to be in the path.

Loading Orcad 386+ library files into the Library Editor is similar. Advanced Schematic Library Editor looks in the current path for a utility called DECOMP32.EXE. Orcad 386+ application is supplied with two utilities DECOMP16.EXE and DECOMP.EXE. DECOMP.EXE is the 32bit library decompiler, this file must be renamed to DECOMP32.EXE, in order for Advanced Schematic Library Editor to automatically decompile and load Orcad 386 binary libraries.

When importing Orcad 386 files, Part field 'Module Name' is converted to 'Text Field 8' in Advanced Schematic.

Graphical Editing Hotkey List

While editing, moving or placing an object you can press the F1 key at anytime to open the Graphical Editing Hotkey List dialog box. This dialog box lists all the hot key commands that can be used while graphically editing, moving or placing an object. This command is context sensitive, therefore the hot listing will change depending on the command and object being editted.

See also

Hot Keys

Save Object Defaults

Advanced Schematic does not have a separate menu or dialog box for defining the current wire size, junction size, text size, etc. Rather, the system remembers the last size, color etc. selected by the user and holds this value until a new value is defined. This also works for objects like rectangles or arcs, whose size and shape is retained during placement. If the Options Preferences Save Defaults option is enabled, these defaults will be saved into your Windows directory in a file called SCH.DFT and LIBEDIT.DFT. The object defaults will be reloaded between editing sessions.

See also

Options Preferences

Communication between Applications

The Protel for Windows system allows the user to launch more than one "instance" of an application or document. For example, you can have two instances of the Schematic Editor running, each dedicated to a different project. When launching another application from the Schematic (sheet) editor, for example, choosing the Run Library Editor command, the outcome is influenced by the number of active instances of the application. If you have a single instance of the Library Editor running, the focus will simply switch to that instance, which then becomes the "active" window. If no instances are running, the program will be both launched and made active. If you have multiple instances running, the application will warn the user that more than one instance of the application is running and no action will occur. Similarly, the Edit button in the Component Browser will display a warning, if more than one instance of the Schematic Editor is running, because the system cannotchoose between open applications.

See also

Run Library Editor
Setup Run Options

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.

Advanced Global Editing For Parts

When Global Editing Parts the contents of a text field can be automatically copied to another text field within the part, (eg. it is possible to copy the contents in Part Field 1 to Part Field 5, or copy the the text in the Part Type field to the Foot Print field). This is done by using special reserved key words to specify the text field that is to be copied. The following is a list of all the fields and corrsponding reserved words that can be used.

Field Name	**Key Word**
Lib Ref	.LIB_REF
Footprint	.FOOTPRINT
Designator	.DESIGNATOR
Part Type	.PART_TYPE
Sheet Path	.SHEET_PATH
Part Field 1	.PART_FIELD_1
Part Field 2	.PART_FIELD_2
Part Field 3	.PART_FIELD_3
Part Field 4	.PART_FIELD_4
Part Field 5	.PART_FIELD_5
Part Field 6	.PART_FIELD_6
Part Field 7	.PART_FIELD_7
Part Field 8	.PART_FIELD_8
Part Field 9	.PART_FIELD_9
Part Field 10	.PART_FIELD_10
Part Field 11	.PART_FIELD_11
Part Field 12	.PART_FIELD_12
Part Field 13	.PART_FIELD_13
Part Field 14	.PART_FIELD_14
Part Field 15	.PART_FIELD_15
Part Field 16	.PART_FIELD_16

Fields in Schematic Library Editor

Default Designator	.DEFAULT_DESIGNATOR
Sheet Part Filename	.DEFAULT_SHEET_PART_FILENAME
Foot Print 1	.FOOT_PRINT_1
Foot Print 2	.FOOT_PRINT_2
Foot Print 3	.FOOT_PRINT_3
Foot Print 4	.FOOT_PRINT_4
Description	.DESCRIPTION
Text Field 1	.LIBRARY_PART_FIELD_1
Text Field 2	.LIBRARY_PART_FIELD_2

Text Field 3	.LIBRARY_PART_FIELD_3
Text Field 4	.LIBRARY_PART_FIELD_4
Text Field 5	.LIBRARY_PART_FIELD_5
Text Field 6	.LIBRARY_PART_FIELD_6
Text Field 7	.LIBRARY_PART_FIELD_7
Text Field 8	.LIBRARY_PART_FIELD_8
Part Field Name 1	.PART_FIELD_NAME_1
Part Field Name 2	.PART_FIELD_NAME_2
Part Field Name 3	.PART_FIELD_NAME_3
Part Field Name 4	.PART_FIELD_NAME_4
Part Field Name 5	.PART_FIELD_NAME_5
Part Field Name 6	.PART_FIELD_NAME_6
Part Field Name 7	.PART_FIELD_NAME_7
Part Field Name 8	.PART_FIELD_NAME_8
Part Field Name 9	.PART_FIELD_NAME_9
Part Field Name 10	.PART_FIELD_NAME_10
Part Field Name 11	.PART_FIELD_NAME_11
Part Field Name 12	.PART_FIELD_NAME_12
Part Field Name 13	.PART_FIELD_NAME_13
Part Field Name 14	.PART_FIELD_NAME_14
Part Field Name 15	.PART_FIELD_NAME_15
Part Field Name 16	.PART_FIELD_NAME_16