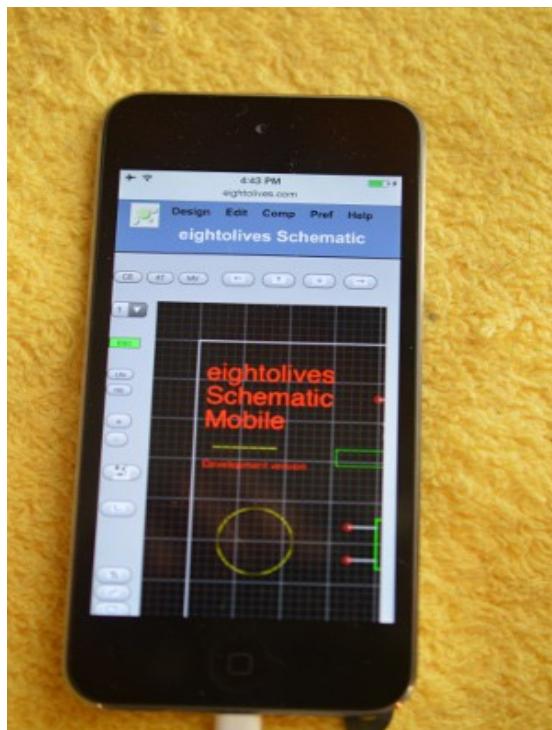


eightolives

Eightolives' Schematic Mobile User's Manual



January 12, 2017

copyright (c) 2017 William M Kaupinis, All rights reserved.

Table of Contents

1 Introduction.....	5
2 Applicable Documents.....	6
3 Overview.....	7
3.1 Basics.....	8
3.2 The Design Process.....	8
3.3 The Design Project Concept.....	8
3.4 Design Guidelines.....	8
3.4.1 Naming Nets.....	8
3.4.2 Connecting Nets.....	9
3.4.3 Naming Components.....	9
3.4.4 Design Checks.....	9
4 Starting the Software.....	11
4.1 Starting the On-Line Version.....	11
4.2 Updating the software.....	11
5 Using Schematic Mobile.....	11
6 Using the Graphical User Interface.....	12
6.1 Menus.....	13
6.1.1 Design Menu.....	13
6.1.2 Edit Menu.....	17
6.1.3 Component Menu.....	20
6.1.4 Preferences Menu.....	21
6.1.5 Help Menu.....	23
6.2 Selected Component Menu.....	24
6.3 Selected Net Menu.....	26
7 Symbols.....	28
7.1 Eightolives On-Line Symbols.....	28
7.2 Connector Wizard.....	28
7.3 Block Symbol Wizard.....	29
7.4 Using Vendor "bzl" Files.....	30
7.5 Manually Drawing Symbols.....	30
7.6 Using gEDA Library Symbols.....	30
7.7 Special Symbols.....	30
7.7.1 VHDL Ports.....	30
7.7.2 Net Stubs or Off sheet references.....	31
7.7.3 Simulatable symbols.....	31
7.7.4 Constants.....	31
8 Attributes.....	32
8.1 gEDA Defined Attributes.....	32
8.1.1 refdes.....	32
8.1.2 device.....	32
8.1.3 netname.....	33
8.1.4 net.....	33
8.1.5 footprint.....	33
8.1.6 pinnumber.....	33
8.1.7 pinlabel.....	33
8.1.8 pinseq.....	33

8.1.9 pintype.....	33
8.1.10 description.....	34
8.1.11 author.....	34
8.1.12 comment.....	34
8.1.13 numslots.....	34
8.1.14 slot.....	34
8.1.15 slotdef.....	34
8.1.16 documentation.....	35
8.1.17 dist-license.....	35
8.1.18 use-license.....	35
8.1.19 symversion.....	35
8.2 Special Attributes.....	35
8.2.1 partid.....	35
8.2.2 padsdevice.....	35
8.2.3 sheetnumber.....	35
8.2.4 netcomment.....	35
8.2.5 labelissignal.....	35
8.2.6 ncok.....	35
8.2.7 model.....	35
8.2.8 primitive.....	36
8.2.9 busname.....	36
8.2.10 size.....	36
8.2.11 limit1.....	36
8.2.12 limit2.....	36
8.2.13 logic_type.....	36
8.2.14 src.....	36
8.2.15 gnd_plane_signal.....	36
8.2.16 power_plane_signal.....	36
8.2.17 footprint.....	36
8.2.18 placenear.....	36
8.2.19 Timing Attributes.....	37
9 File Formats.....	38
9.1 Concatenated Schematic Files.....	38
9.2 File names.....	38
9.3 Version.....	38
9.4 Coordinate Space.....	38
9.5 Colors.....	38
9.6 Drawing Objects.....	39
9.7 Line.....	39
9.8 Box.....	40
9.9 Circle.....	41
9.10 Arc.....	41
9.11 Text and Attributes.....	42
9.12 Net.....	43
9.13 Pin.....	43
9.14 Component.....	44
9.15 Bus.....	44
9.16 Picture.....	44
9.17 Path.....	45

9.17.1 Path Data.....	45
10 Compatibility Exceptions to gEDA.....	47
11 Services.....	48
11.1 Local Service.....	48
11.2 Server Service.....	48
11.3 External Service.....	48
11.4 Drive Service.....	49
11.5 A Computer's File System.....	49
12 Schematic Mobile Server Service Interface.....	49
12.1 Base URL.....	49
12.1.1 projects.rss.....	49
12.1.2 Project Directory.....	50
13 Feedback.....	50

1 Introduction

This document describes the use of the eightolives Schematic Mobile software program. Schematic Mobile is a web-browser-based Graphical User Interface (GUI) for creating electrical schematics. Schematics, symbols and netlists are compatible with gEDA formats which are ASCII text files that can be humanly edited. The Javascript-based software works in PC environments or in mobile environments such as Apple's iPod Touch. The software is tested with Firefox and Safari browsers.

2 Applicable Documents

gEDA Project Wiki	http://wiki.geda-project.org
gEDA/gaf File Format Document	http://wiki.geda-project.org/geda:file_format_spec
gEDA Attributes List	http://wiki.geda-project.org/geda:master_attributes_list

3 Overview

Eightolives' Schematic Mobile is a HTML/Javascript-based web app used for manual generation of electrical schematics. Schematic, symbol, and netlist formats are compatible with the gEDA formats. A user's view of the tool is shown in Figure 1.

The major functions you do with the tool include:

- Manually generate or read electrical schematics in gEDA format.
- Use built-in symbols, web-based eightolives' symbols, gEDA symbols (not provided) or manually create your own symbols within the tool.
- Perform design checks.
- Save schematic and symbols.
- Create a netlist in gEDA, PADS, KiCad, VHDL or eightolives' Hardware Model format for use by other tools.
- Create a Bill of Materials.
- Interface with Google Drive or an optional external server for remote file storage and additional library support.
- Start and interact with a WaveformViewer simulation or eightolives' Layout tool.

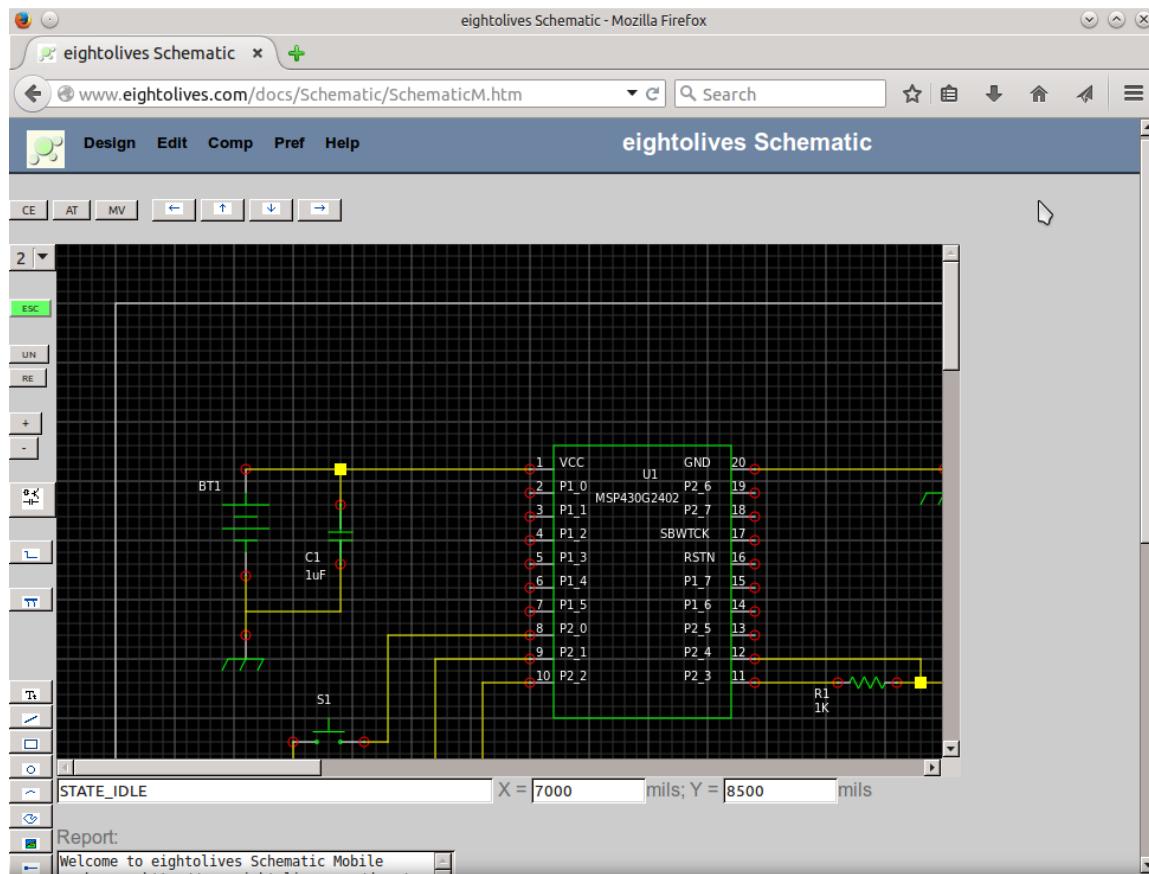


Figure 3-1 eightolives' Schematic Mobile Screen Shot

3.1 Basics

A schematic is a graphical representation of an electrical circuit. It is typically used to specify the circuitry of a printed circuit board (PCB) or subsystems that involve a variety of different components. It can also be used to capture a design for simulation or just provide a graphic for documentation.

You create a design by placing or instantiating **symbols** that represent the parts being used. These symbols have *pins* that represent the electrical input and output signals to which you connect wires or **nets**.

The interconnect information of all the nets is called a **netlist** which is an output file. The netlist is used by other tools to create printed circuit board artwork, simulation databases or design checking and test aid.

3.2 The Design Process

Capturing a design schematic is just part of the process to develop a printed circuit board or subsystem. A block diagram using generic, “black-boxes” is a form of schematic developed early in the design cycle to capturing a high level partition of functionality. As a design is refined in more detail, parts are identified and symbols can be selected from a library or created and stored in the project's library.

The unit's inputs and outputs, once identified, can be assigned to connectors which are also represented by symbols. When all pins have been connected to nets, design checks can be run to identify errors, potential problems

3.3 The Design Project Concept

A design project is considered to consist of a set of schematic sheets that represent a design and are located in a defined directory structure that includes the symbols used. A “service” manages the storage of data in a particular place.

The LOCAL project is stored in your browser's cache memory. You must not clear the cache or off-line data area if you want to always have access to data in this project. The LOCAL service is the default selection. It only supports a single, LOCAL project.

When you run Schematic Mobile from the eightolives site, you can select the Server service which lets you select various example demonstration projects. These projects are read-only, but you can save the files in LOCAL project or in your computer's file system.

External service lets you access projects and other libraries on the same or different computer. Currently a computer running eightolives Microserver on your local network can act as a project server allowing you to access and save data to projects that you can establish and manage.

Drive service lets you use your Google Drive account for storage.

3.4 Design Guidelines

3.4.1 Naming Nets

Nets are named using the “netname” attribute. It is recommended that net names begin with an alpha character making it compatible with VHDL signal names. Signal names should only consist of alpha and numeric characters and the underscore character. Power signals should not start with a “+” or “-”

character, use “P” or “M” instead. Use the letter “R” instead of a “.”. Recommended power signal names include P5V, M12V, P3R3V, VCC, VDD.

When creating nets, the tool creates a name of the format “N_xxxxxxx” where x are digits. Renaming nets to meaningful names can help in circuit debug.

3.4.2 Connecting Nets

Nets are usually drawn as “wires” connecting signal pins on a sheet. Net names are common between all sheets of a schematic project.

Component pins can connect to component pins such as a resistor pin at the same location as a capacitor pin will assume connection by a net.

A special eightolives component called an “offsheets reference” or “net stub” consists of a pin and arrow graphic that can be used to connect a component pin to a net without explicitly drawing a net “wire”. This is also used to connect pins to nets on other sheets. You set the associated net by editing the offsheet reference's net attribute to specify the appropriate net name or by editing using a component's pin editor.

(The pin of an offsheet reference has the special eightolives attribute “labelissignal” which automatically assigns the visible pinlabel to be the name specified by the net attribute.)

3.4.3 Naming Components

Physical parts that make up a circuit are represented by a Component that has a “refdes” and “device” attribute.

The refdes attribute uniquely identifies a part. It usually consists of an upper case letter(s) and incrementing number. Examples are: R1 for a resistor, C12 for a capacitor, Q3 for a transistor, U1 for an integrated circuit, A4 for an arbitrary subassembly. Library symbols normally include the recommended default letter.

Lower case letters at the end of a refdes indicate that the associated symbol represents just part of the complete component. For example, U1a might be used for one logic gate symbol when there may be four gates in the physical component with U1b, U1c and U1d representing the other three. Similarly, packages with large pin counts such as FPGAs may be represented by several smaller symbols to better fit on a schematic sheet or span several sheets.

The device attribute identifies the part and should start with a valid VHDL name, starting with an alpha character, and consisting of alpha, numeric or underscore characters with no spaces. Examples include: RESISTOR_330_0603, Q2N3906, SN74HCT00,

Components destined for circuit board layout should have a “footprint” attribute added to specify the “artwork” symbol used by the circuit board layout tool.

Other arbitrary attributes can be used to specify other information such as “partnumber” and “inventorynumber”.

3.4.4 Design Checks

You can perform a check of your design by selecting the menu option Design > Check Design. This

provides an editor window reporting results of various component and net checks. Errors and warnings are reported as well as observations which may or may not be relevant. The user should review all comments to ensure that each observation is acceptable for the design.

An additional check can be performed by creating and reviewing a netlist. Scanning a netlist can sometimes provide revelations that might affect the design.

4 Starting the Software

4.1 Starting the On-Line Version

You start the on-line version of the tool by navigating your web browser to:
<http://www.eightolives.com/docs/Schematic/SchematicM.htm>

The link is also available from the eightolives Home Page, the Schematic Mobile web page or from the eightolives QuicKApps menu.

The browser should be configured to run Javascript and allow off-line (cookie) data storage.

4.2 Updating the software

The on-line version is periodically updated. When you access the tool from the eightolives server, you get the latest version.

5 Using Schematic Mobile

Schematic Mobile uses mouse or finger actions to select, instantiate and operate on the drawing objects and design. Single left click, or single touches are the primary initiating events.

When an object is selected, a bounding box appears on screen. Mode or state information may be displayed at the bottom of the window. Clicking or touching outside a selected area normally unselects all objects and returns to Idle Mode.

Objects are normally saved only when the user specifically saves the schematic sheet or symbol.

The location of visible attributes can be changed by editing that attribute and setting its x and y values. Some visible attributes such as refdes can be moved by activating an option on the component's popup menu.

When you open a schematic sheet, previously opened sheets are retained.

A typical opening sheet sequence is:

- Start Schematic Mobile
- Design > Open Project, select a service, select a project,
- Design > Open Project File, select a “.sch” file to load,
- Design > Delete Sheet 1 to delete the blank sheet 1 that was retained
- Design > Fix Refdes to attempt to resolve any open refdes issues,
- Design > Update Nets to attempt to resolve any interconnect issues,
- Design > Check Design to see if there are errors or warnings detected.

6 Using the Graphical User Interface

The Schematic Mobile window is shown in Figure 6-1.

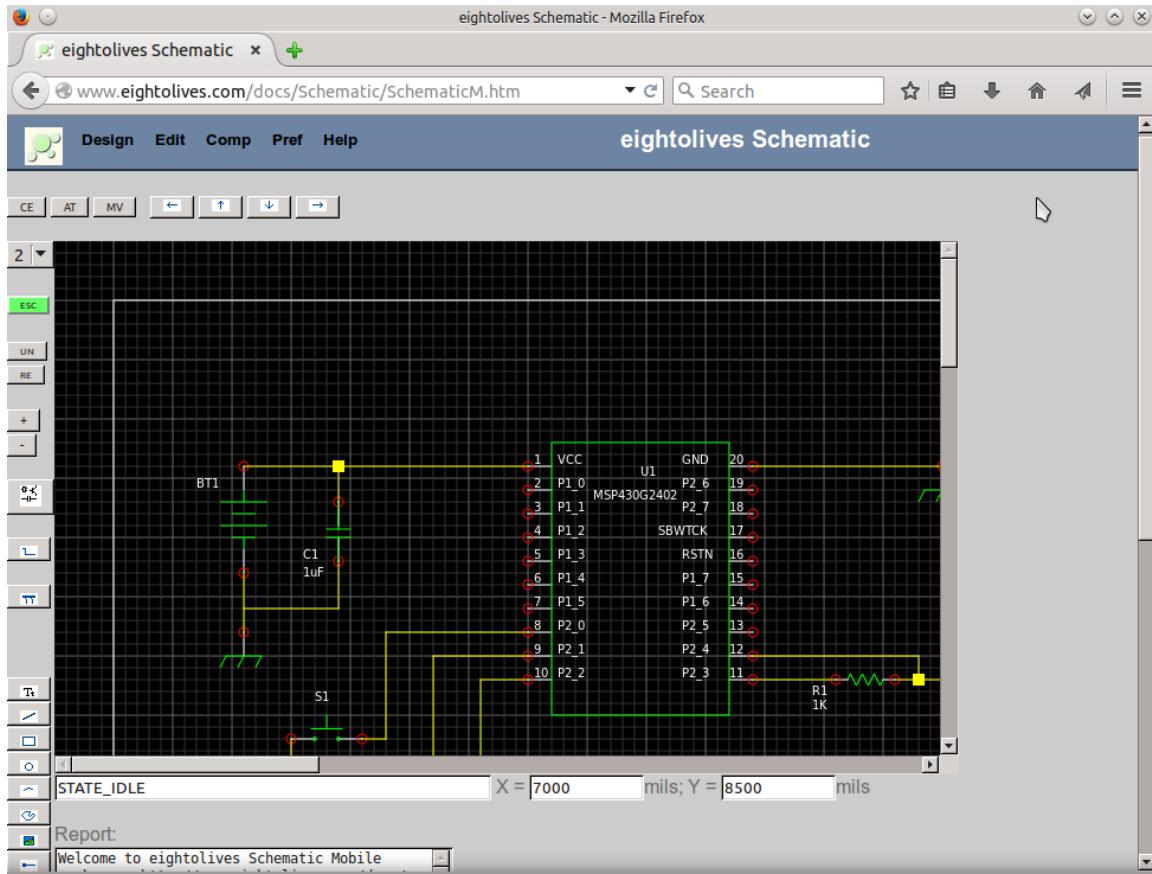


Figure 6-1 eightolives Schematic Mobile

The top of the window has menu items. Below the menu options are a few buttons for component editing (CE), attribute editing (AT), move a selection (MV), and fine moving arrows for the selected item. The left side has the sheet selector, escape button (ESC), Undo and Redo buttons, zoom buttons (+ and -), the Component Sector button, the draw net button and draw bus button. Additional buttons are used for drawing graphics or symbols. Those buttons create attributes, text, lines, boxes, circles, arcs, arbitrary shapes, pins and pictures.

The default sheet border template is A size. You can disable the sheet border from the Pref menu. The actual drawing universe is 11.5 inches by 8.5 inches. A Pref menu option, Auto-offset, is normally enabled. This ensures that A size schematics created with other tools will display correctly in this A size drawing universe.

Left mouse clicks or touches are used to initiate most actions. A click is used to select a component or net. A second click or touch displays a popup menu of options. A click away from a component or net returns the tool to Idle State.

A click of a component's pin, initiates the drawing of a net from that pin. A second click of the pin displays its popup menu.

The bottom panel shows state, status and mouse coordinates in mils (thousandths of an inch).

The Report Window displays various status, debug information and generated reports.

6.1 Menus

6.1.1 Design Menu

The Design Menu options are shown in Figure 6-2

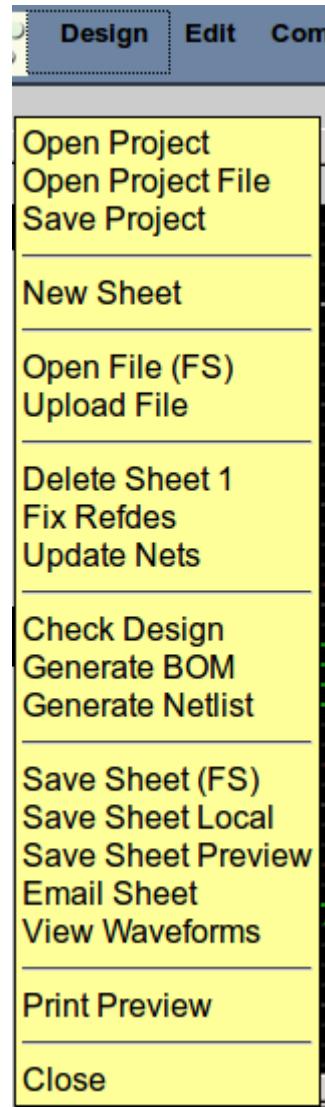
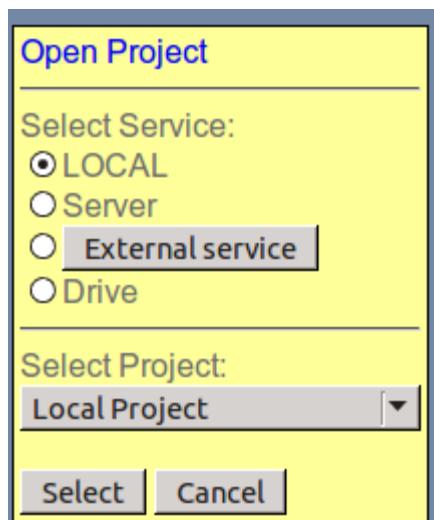


Figure 6-2 Design Menu Options

Open Project

The Open Project option lets you select a service and a project associated with that service. The default service is LOCAL which uses the browser's "cookie" cache memory or "localStorage" for saving and retrieving files. The LOCAL service only supports LOCAL Project.

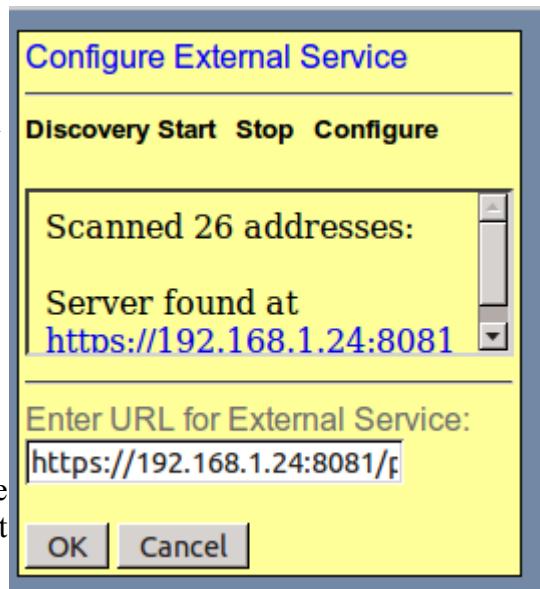
The Server service uses available resources from the server where you are retrieving Schematic Mobile. For the on-line version, it's the eightolives web server which offers read-only, demonstration projects and various



eightolives symbol libraries.

The External service lets you use storage facilities on an external server. It's intent is to access another project server or "cloud" service while using the on-line version. Currently it works with eightolives Microservers that are active on your local network.

Clicking the External Service button in the Open Project window pops up the Configure External Service Window. You can enter the URL of the external service and click OK. To automatically find Microservers on your network, click Discovery Start which will scan 256 local addresses. Discovered servers are listed. You click on the discovered server link to set the URL for that server and then click OK.



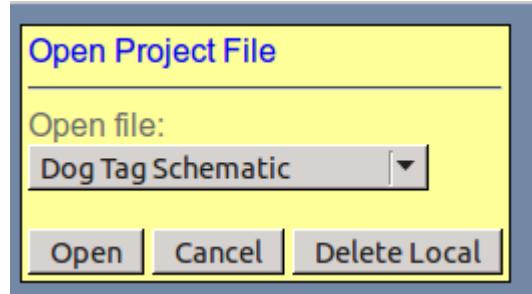
The Drive service connects to your Google Drive account. When you select Drive, Google support software is loaded. If you are not logged in to Google Drive, the Google log-in window will appear. After successful login, the words "Google Drive" appear in the Select Project box. Currently, Google Drive is treated as a single project much like Local Project. Note that when you use Google Drive, you are subject to their terms and conditions.

Once you select a service you can select an available project listed in the Open Project window and access resources available for that project.

Open Project File

The Open Project File option displays a drop-down menu list of available files which you can open.

When LOCAL project is selected, you also have the option of deleting the local file.



Save Project

The Save Project option will save all current schematic sheets and symbols to the selected project area. When using Schematic Mobile from the eightolives site, you do not have Save Project privileges to the eightolives Server service so you are limited to saving only to LOCAL project or External service.

See the other Save options for saving individual schematic sheets or symbols.

New Sheet

The New Sheet option opens an additional sheet in the current project.

Open File (FS)

The Open File (FS) option lets you select a schematic or symbol using the browser's File Chooser window. This option is only available on platforms that support File System (Apple's iPod does not).

Upload File

The Upload File option lets you select a file using the browser's File Chooser window which is then uploaded to your selected project area. This requires that your platform supports File System and that you have upload privileges to the selected project.

Delete Sheet 1

When you open a sheet, it is added to whatever schematic sheets are already opened. A frequent operation after loading the first sheet is to delete the previous sheet 1.

Fix Refdes

The Fix Refdes option will attempt to assign consecutive numbers to unassigned component reference designation attributes (refdes).

It will not assign refdes numbers on “slotted” symbols, symbols that represent part of a component. Check Design reports on any unassigned refdes issues.

Update Nets

The Update Nets option attempts to correct the net naming of various drawn net segments in the schematic. It also adds “junctions” where related net segment connect.

Update Nets should be executed often to ensure consistency of the net names. Check Design reports on net issues.

Check Design

The Check Design option executes various consistency checks and generates a Check Design Report that appears in the Report window and in a separate popup on certain platforms. Check Design checks and reports on various net and bus issues and then checks each schematic sheet for refdes issues and component connections.

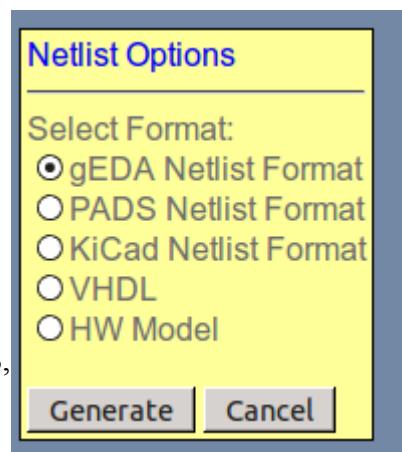
Check Design does not establish the “goodness” of a design or whether it will do anything if realized. It only checks for obvious defects or possible issues relating to connectivity.

Generate BOM

This option creates a Bill of Materials (BOM), a list of components used in the design. The list includes quantity, the component's “partid” attribute or component name and applicable refdes assignments. The BOM appears in the Report window and in a separate popup on certain platforms.

Generate Netlist

The Generate Netlist option lets you select which format to use. The gEDA format is the default. Other options are PADS, KiCad, VHDL or eightolives Hardware Model. The netlist appears in the Report window and in a separate popup on certain platforms.



Netlists intended to be used for printed circuit design require all intended connections be nets made to pins. Buses should only be used for block diagrams or VHDL output designs.

VHDL output can be used for simulation with appropriate symbol models or can be input into the eightolives Workspaces tool for additional analysis.

The HW Model is a Javascript netlist that can be loaded into eightolives WaveformViewer for defining timing diagrams or for simulation.

Netlists should be manually checked to ensure correctness.

Save Sheet (FS)

The Save Sheet (FS) option lets you save the current sheet to your computer's File System where supported. Schematic sheets should be saved with a ".sch" suffix.

Save Sheet Local

This option saves the current sheet to the LOCAL project. Schematic sheets should be saved with a ".sch" suffix.

The Open Project Symbol lets you select a symbol from your project's symbol directory.

Save Sheet As

The Save Sheet As option lets you save the current schematic sheet.

Save Sheet Preview

The Save Sheet Preview option displays the sheet's ".sch" text file data in the Report window and in a separate popup on certain platforms.

Email Sheet

This option attempts to open your platform's email form with the ".sch" text file data listed in the email's body.

View Waveforms

This option will create an eightolives Javascript Hardware model of the schematic and attempt to load it in a new WaveformViewer window. If successful, then nets can be selected to view its current simulation value or to be added to the WaveformViewer display.

Print Preview

The Print option lets you view what would be printed on a system printer. Printing is handled by your browser's capabilities.

Close

The Close option lets you close the menu.

6.1.2 Edit Menu

The Edit menu is shown in Figure 6-5.

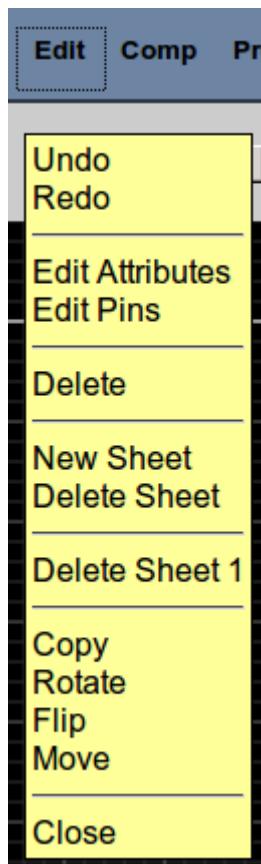


Figure 6-5 Edit Menu

Undo

The Undo option undoes the last graphic command implemented.

Redo

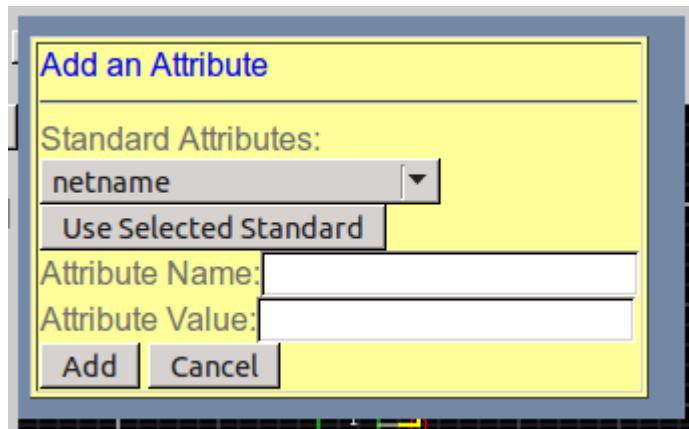
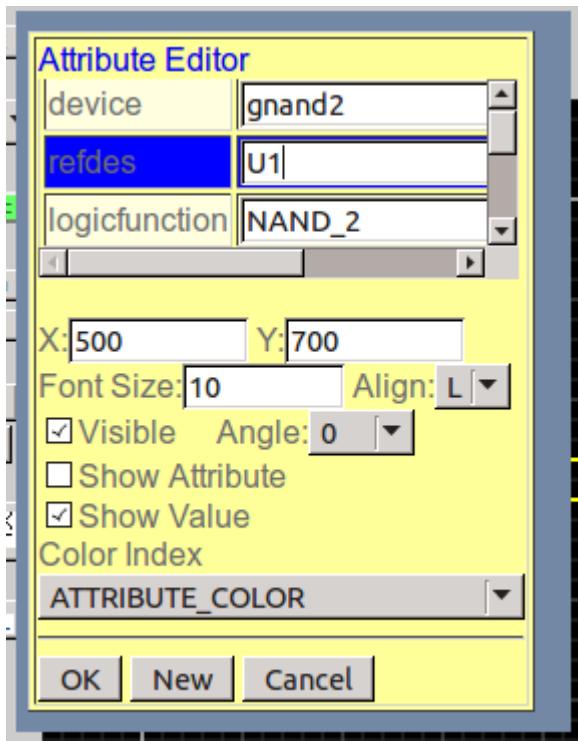
The Redo option reasserts the last “undo” operation.

Edit Attributes

The Edit Attributes option opens the Attributes Editor window for the currently selected object. You can also use the “AT” button to open this window.

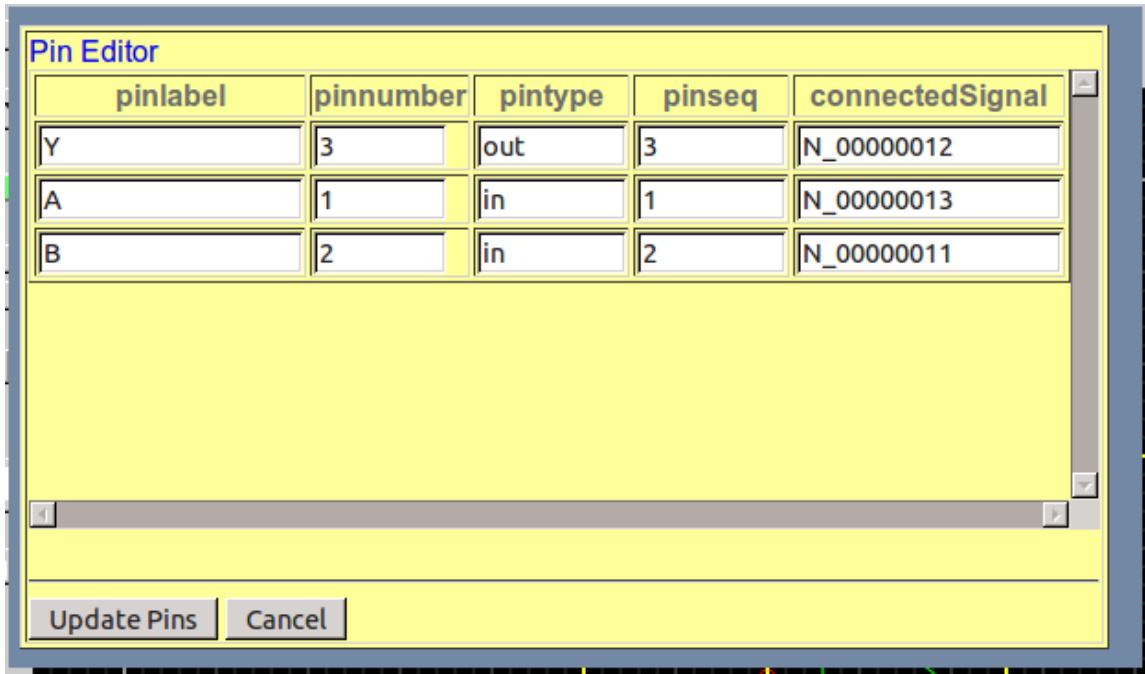
You can select an existing attribute and edit its value or characteristics. Click OK for the changes to take effect.

To add a new attribute, click the New button and the Add an Attribute window appears. You can select a standard Attribute Name or just enter one in the text field. Enter the Attribute Value and then click the Add button.



Edit Pins

The edit Pins option lets you edit pin and connection information of a selected component.



Delete

The Delete option lets you delete the selected object after conformation.

New Sheet

This option adds a new sheet to the schematic.

Delete Sheet

This option lets you delete the current sheet after conformation.

Delete Sheet 1

This option lets you delete sheet 1 after conformation. It is useful as existing sheets are retained when a new schematic is opened.

Copy

The Copy command creates a new instance of the selected component. The symbol source file is read to create the copy.

Rotate

The Rotate option rotates the symbol around its origin.

Flip

The Flip option flips the symbol around the vertical axis.

Move

The Move command lets you move the selected item. It is the same as clicking the "MV" button.

Close

This option closes the menu.

6.1.3 Component Menu

The Component menu is shown in Figure 6-8.

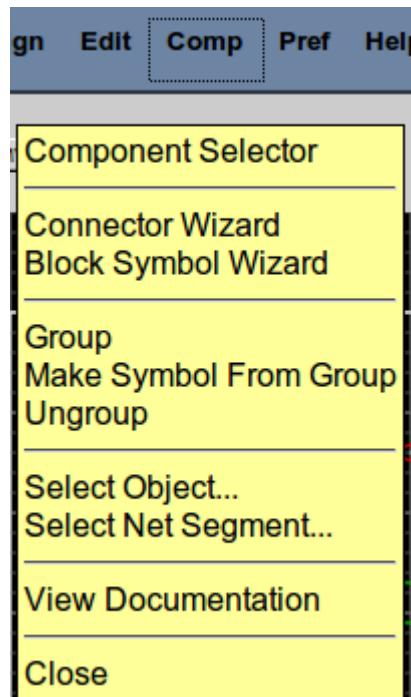
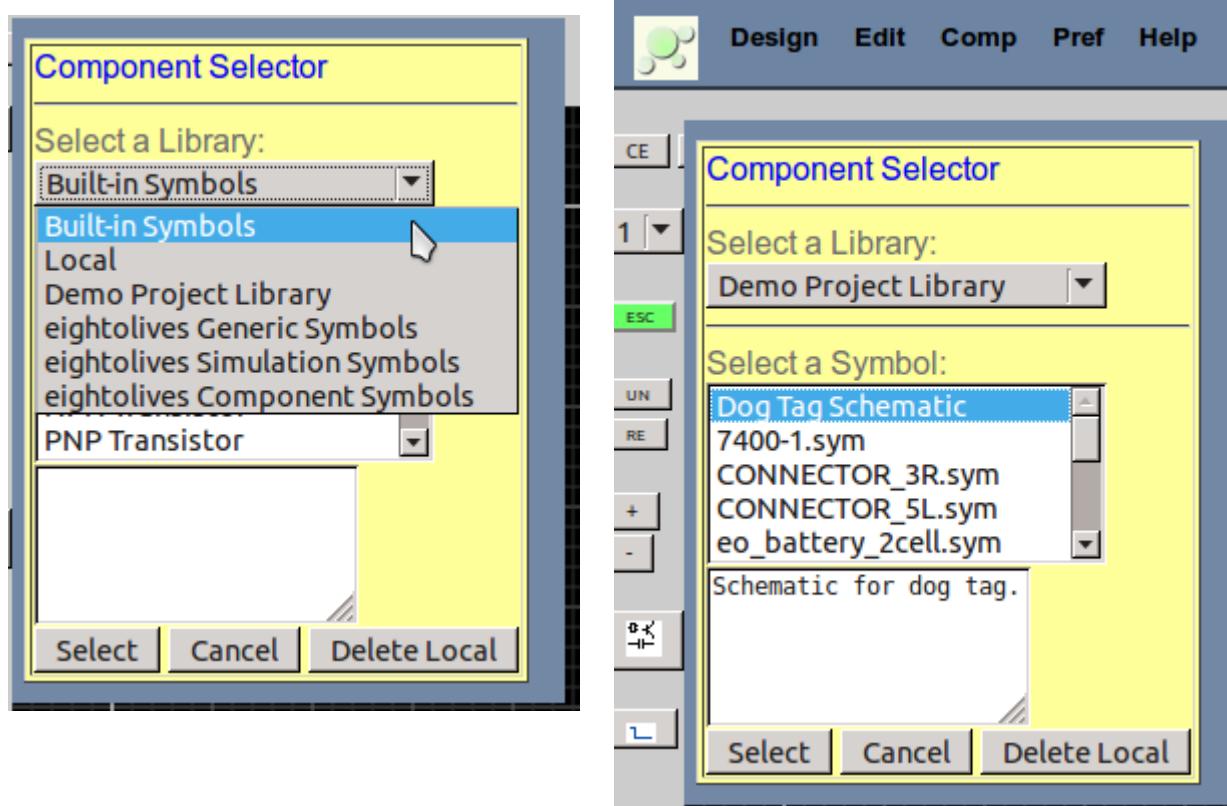


Figure 6-8 Component Menu

Component Selector

The Component Selector opens the Component Selector Window. From this window you can select symbols from the built-in, LOCAL, project libraries and eightolives on-line libraries.



Connector Wizard

The Connector Wizard option lets you create generic, multi-pin connector symbols or dual-row header symbols. You are prompted for the number of pins, the refdes (e.g. J6) and whether the pins should break right or left.

Block Symbol Wizard

The Block Symbol Wizard option lets you create black-box, block-type symbols that can have pins on all four sides. The wizard window lets you specify refdes and device attributes and the number of pins on each of the four sides. A half-size option is available for large pin count devices. Pin numbers are assigned counterclockwise from the upper left corner.

Group

The Group option lets you draw a rectangle around a group of drawing objects. Grouped objects can be moved or made into a symbol.

Make Symbol from Group

The Make Symbol from Group option lets you create a symbol from a group containing graphic elements and pins to form a symbol. Symbols cannot contain components, nets or bus elements. You are prompted for refdes and device attributes.

Ungroup Symbol

The Ungroup Symbol option deconstructs a symbol into its individual drawing objects. Component attributes are lost in the process.

Select Object

The Select Object option allows you to select a component from a menu list.

Select Net Segment

The Select Net Segment option lets you select a net segment from a menu list.

View Documentation

This option will open a new browser window if the selected component has the documentation attribute specified. The attribute's value is the full URL to the documentation.

6.1.4 Preferences Menu

The Preferences menu is shown in Figure 6-9.

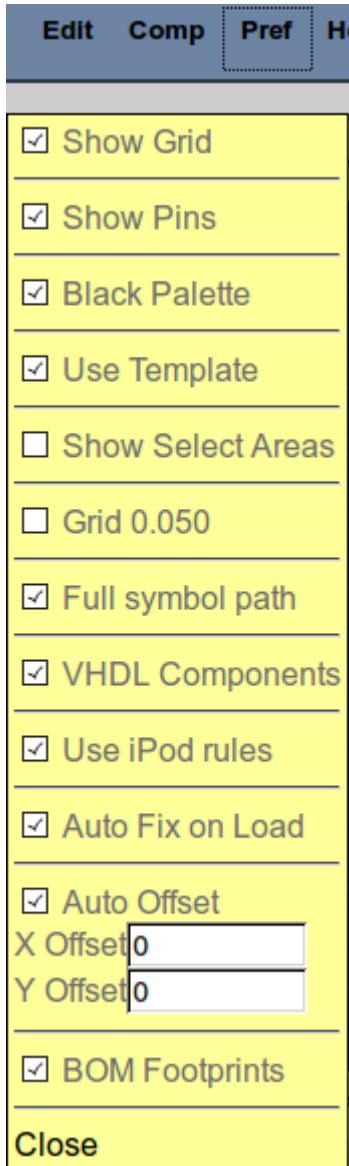


Figure 6-9 Preferences Menu

Show Grid

The Show Grid option displays the grid..

Show Pins

The Show Pins option displays a component's pins as red circles. This is useful to identify where to connect a net, but may not be desirable for printing.

Black Palette

This option displays the schematic with a black background. Unchecking results in a white background.

Use Template

This option displays the default A size sheet border template when a new sheet is created.

Show Select Areas

This option displays all component select boxes. This is useful to help see the size and location of the select boxes.

Grid 0.050

This option lets you specify a 0.050 inch grid. The 0.1 inch default setting is recommended. The 0.05 inch setting may be needed when using certain gEDA symbols or in creating certain custom symbols.

Full Symbol Path

This option saves the full symbol path to the source .sym file. The gEDA format only saves the filename. The full path may alternatively be specified in the eightolives attribute “src”.

VHDL Components

This option includes component declarations in the generated VHDL output.

Use iPod Rules

This option modifies certain tool behaviors because of iPod constraints (primarily inhibiting certain popup windows).

Auto Offset

This option adjusts the location of the schematic sheet to be within Schematic Mobile's A size display universe. A size schematics created with other tools may be located elsewhere in a larger display universe. When not checked, the X Offset and Y Offset text values determine the offset to be used.

BOM Footprints

This option , when checked, adds the footprints assigned to the generated BOM report.

Close

This option closes the menu.

6.1.5 Help Menu

The Help menu is shown in Figure 6-10.

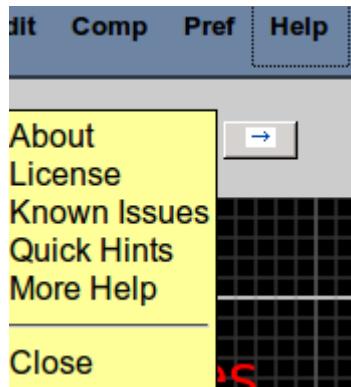


Figure 6-10 Help Menu

About

The About option displays the version information for the tool.

License

The License option displays the tool's license.

Known Issues

A list of known issues is opened in a new browser window.

Quick Hints

This option opens the Quick Hints web page which summarizes essential information on using the tool.

More Help

The More Help option opens the Schematic Mobile web page which has links to manuals, papers and other information.

Close

This option closes the menu.

6.2 Selected Component Menu

The Selected Component Menu appears when you select a component and then click it again. The menu is shown in Figure 6-11.

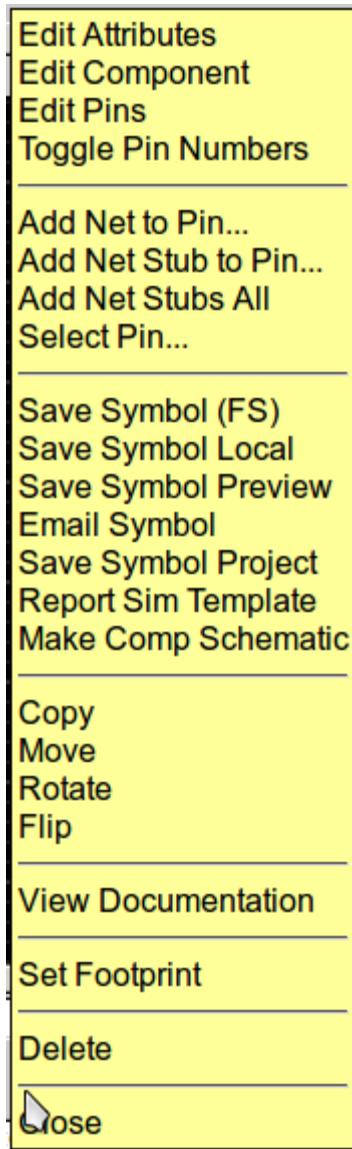


Figure 6-11 Selected Component Menu

Edit Attributes

This option opens the Edit Attributes Window.

Edit Component

This option opens the Edit Component Window.

Edit Pins

This option opens the Edit Pins Window.

Toggle Pin Numbers

This option can display or hide the pin numbers of the component.

Add Net To Pin

This option lets you select a pin number and then starts a net placement from that pin.

Add Net Stub To Pin

This option lets you select a pin number and then places a net stub on that pin.

Add Net Stubs All

This option places net stubs on all pins that don't have an existing connection.

Select a Pin

This option lets you select a pin from a select table.

Save Symbol (FS)

This option lets you create and save the symbol file in your File System if available.

Save Symbol Local

This option creates and saves the symbol file in the Local Project.

Save Symbol Preview

This option displays the symbol text file in the report window and in a popup in some systems.

Email Symbol

This option attempts to email the symbol text file.

Save Symbol Project

This option attempts to save the symbol text file in the selected project.

Report Sim Template

This option creates an eightolives Hardware Model Template for the component and displays it in the report window. This model is used for simulation in WaveformViewer. You can copy, edit or save it from the report window.

Make Comp Schematic

This option creates a schematic template of the selected component. This schematic consists of the component's I/O ports. You can use this to help create a Hardware Model of a component that consists of some existing Hardware Models by instantiating and connecting these sub-components and then creating a Hardware Model template using the Design > Generate Netlist > HW Model menu option.

Copy

This option reloads the symbol's source file creating a copy of the symbol for placement.

Move

This option lets you move the symbol.

Rotate

This option rotates the symbol 90 degrees.

Flip

This option flips the symbol.

View Documentation

This option opens a new browser window if the symbol's documentation attribute is specified with a URL to appropriate documentation.

Set Footprint

This option provides a popup menu from which you can select a gEDA footprint for the selected part.

Delete

This option deletes the component upon confirmation.

Close

This option closes the menu.

6.3 Selected Net Menu

The Selected Net menu is shown in Figure 6-12

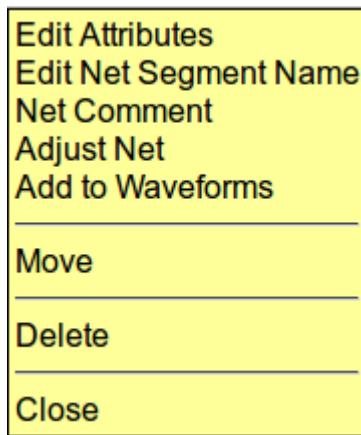


Figure 6-12 Selected Net Menu

Edit Attributes

This option opens the Edit Attributes window.

Edit Net Segment Name

This option lets you change the netname attribute for this segment.

Net Comment

This option lets you add a comment to the net.

Adjust Net

This option lets you adjust a net using the arrow buttons.

Add to Waveforms

This option adds this signal to the associated WaveformViewer signal display. The WaveformViewer must have been opened using the Design > View Waveforms command.

Delete

This option deletes this net segment upon confirmation.

Close

This option closes the menu.

7 Symbols

Symbols are graphical symbols that represent a part. Symbols are composed of the graphic that is visible on the schematics, pins which represent each electrical connection to the part, and attributes which defines other characteristics of the part.

To represent a real part for printed circuit layout on a schematic, all inputs, outputs and power connections must be specified by the symbol. Generic symbols may be appropriate for simple devices such as resistors and capacitors, but symbols specific to a inventory part probably requires a unique symbol for each inventory part used.

Symbols can be obtained from the eightolives, on-line library which supplies basic symbols and symbols for some parts used by eightolives.. Symbols generated for the gEDA project can also be used. The Schematic tool also has “wizards” which let you easily create generic connector symbols and header symbols of different sizes or block-type (black box) symbols with pins possible on all four sides.

You can also create a symbol by drawing the graphic, adding pins, and grouping the items to create the symbol. You can edit symbols by editing the symbol's text file. You can edit various attributes

7.1 Eightolives On-Line Symbols

The eightolives symbols are accessed from the Component Selector Window which you open from the menu *Comp > Component Selector* or using the left column Component Selector button.

The eightolives Symbols folder contains generic symbols. After placing a symbol you may need to edit its Attributes by selecting the component and the clicking the “AT” button.. You typically edit the refdes attribute to specify the instance (e.g. C12) and the device attribute which is used as the filename in saving the symbol to your project. Resistors and capacitors also have a value attribute which specifies the components specific value.

The eightolives Simulation folder contains symbols that correspond to simulatable elements found in the eightolives Simulation Library. This includes basic logic gates, flip flops and other basic functions such as clock, counters and delay. A schematic created with these elements can be saved as a VHDL file and then loaded into Workspaces Desktop's Design Tool and then simulated using the WaveformViewer.

The eightolives Parts folder contains symbols of certain parts created and used by eightolives.

When you save your schematic and symbols to your project directory, you will then use the saved version of the symbol.

7.2 Connector Wizard

The Connector Wizard creates generic multi-pin connector symbols or dual row header symbols for your schematic. You start the wizard from menu option *Comp > Connector Wizard*. Three dialog windows ask you to specify the number of pins, the refdes attribute (e.g. J3) and whether the pins should go RIGHT or LEFT. For headers specify the number of header pins (which must be an even number) and select whether the pins are on the right, left or both sides of the symbol.

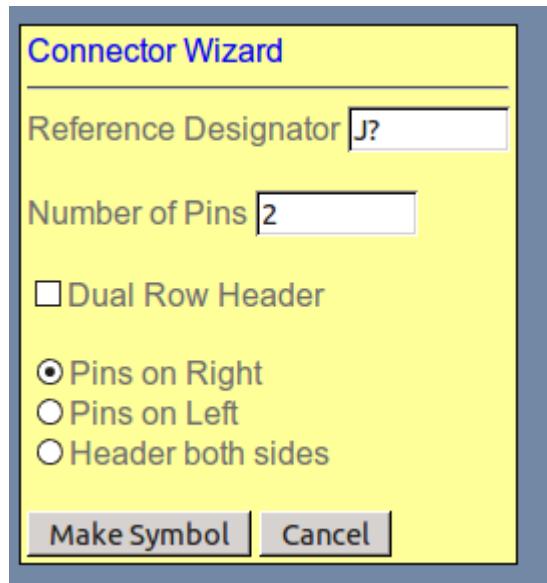


Figure 7-1 Connector Wizard

7.3 Block Symbol Wizard

The Block Symbol Wizard creates a box type symbol. You start the wizard from menu option *Comp > Block Symbol Wizard*. The wizard is shown in Figure 7-2.

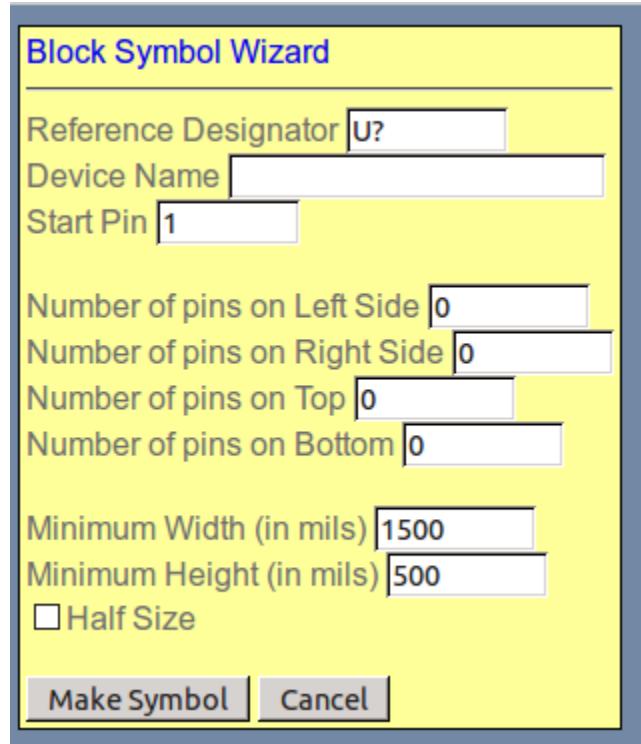


Figure 7-2 Block Symbol Wizard

You specify the refdes, device name, and the number of pins that should appear on each side of the symbol. Half size can be used to create a high pin count symbol that will fit on a small sheet such as a 144 pin QFP which has 36 pins on each side.

The created symbol assigns pin numbers counter-clockwise from the upper left corner. Default pin labels are the letter “P” followed by the pin number. Once you instantiate the component, click the component and select Edit Pins to customize pin labels and pin types.

Use the Edit Attributes window to edit attribute values, make them visible or invisible, or set their display location.

Save the symbol to your project directory.

7.4 Using Vendor “.bxl” Files

You can indirectly import symbols that are provided by vendors as “.bxl” files. These proprietary format files contain both symbol and footprint data in binary format. First, using the free, Ultra Librarian Binary Reader, read the “.bxl” file. Next, export it with “XML Library Format” selected. This creates a “xml” file which can be read by Schematic Mobile.

The created xml file is stored in the “UltraLibrarian\Library\Exported” directory. The stored file name consists of a date stamp with “.xml” suffix.

Opening the xml file with Schematic Mobile produces the symbol which you can place. When you save the symbol using Schematic Mobile, the recommended filename is “*device_name.sym*”. Footprints associated with the symbol are stored in Local project cache. If you wish to use the footprints, you can open them using eightolives’ Editor and then save them.

7.5 Manually Drawing Symbols

You can create symbols by drawing your own. You create the graphic using the Box, Line, Circle, Arc, Text, Path, Picture and Attribute buttons. You then place pins by selecting the Pin button. You then use the menu option *Comp> Group* which lets you select all parts of the symbol and then *Comp > Make Symbol From Group* to create the symbol which you must then save.

This method requires you to edit pin attributes to assign pin numbers and pin labels and control visibility and position. Similarly the component’s refdes and device attributes may need to be edited for value, visibility and position.

7.6 Using gEDA Library Symbols

You can load gEDA “.sym” files if they are in an accessible library from your selected service or you can load them using the menu option Design > Open File (FS) if they are installed on your platform. It’s recommended to have copies of these symbols saved in your project to ensure access.

7.7 Special Symbols

7.7.1 VHDL Ports

The eightolives on-line symbols Input Port (VHDL_INPUT_PORT), Input/Output Port (VHDL_INOUT_PORT) and Output Port (VHDL_OUTPUT_PORT) are special symbols used to define nets that should appear as ports when a VHDL file is created from the schematic. For other netlist tools, these ports will appear as one-pin components similar to hardware test points.

7.7.2 Net Stubs or Off sheet references

The eightolives on-line library has four off-sheet reference components. These are used to connect a net to a component to a component on another sheet. You instantiate the reference, select the component, and edit its net attribute to specify the net to which it connects. Its pin can then be connected to the pin of another component.

7.7.3 Simulatable symbols

Symbols that are to be used to create VHDL files for simulation need to have additional attributes. The *primitive* attribute indicates that there is no structural hierarchy below the block represented by this symbol. The primitive attribute should be specified with the value *true*.

If the symbol is to be associated with a functional library model such as one of the eightolives simulation models, then the *model* attribute must be specified. A value of *simlib* specifies that the eightolives on-line simulation library is the original source of the model.

7.7.4 Constants

The eightolives' sim library has some symbols which can represent constants in VHDL. These include Logic 1, Logic 0, Logic H and Logic L.

8 Attributes

8.1 gEDA Defined Attributes

8.1.1 refdes

The refdes component attribute is the unique reference designation for a part. It normally consists of one or more upper case letters and a sequential number for that letter group. The industry prefix convention is listed in table x.

Device Type	Prefix Letter(s)
Resistor	R
Capacitor	C
Inductor	L
Transistor	Q
Diode	D
Integrated Circuit	U
Switch	SW
Assembly or sub-Assembly	A
Lamps or LEDs	DS
Relays	K
Connectors	J (female) or P (male)
Transformers	T
Batteries	BT
Fuse	F
Quartz crystal	Y
Vacuum tube	V
Meter	M
Jumper	JP
Filter Assembly	FL

For example, a circuit with 10 resistors would have each resistor with a unique refdes value from “R1” to “R10”.

In cases where a symbol refers to only a piece of a complete part, the refdes attribute is given a suffix (lower case) letter. For example, a quad NAND gate part may be represented on a schematic as four separate logic gates with refdes attributes of U1a, U1b, U1c and U1d. The supporting netlist tools will combine the four pieces into one part, U1, discarding the suffix letter.

8.1.2 device

The device component attribute is the name of the symbol file without the “.sym” suffix.

It is useful to have a meaningful name so that the symbol can be identified easily by filename. This attribute should have no space characters and it is recommended that it follow VHDL naming conventions, i.e. starting with an alpha character with the rest being alpha, numeric, underscore characters.

8.1.3 netname

The netname net attribute associates a signal name with a net segment (net) graphic. Nets anywhere on the sheets of a schematic project are electrically connected together. It is recommended that netnames follow VHDL naming conventions, i.e. starting with an alpha character with the rest being alpha, numeric, underscore characters. This would preclude netnames such as “+5V” in favor of “V5V” or “P5V”.

8.1.4 net

The net component attribute connects “hidden pins” on a symbol to nets in the schematic. For example, this attribute can be used to connect power and ground pins of a part represented by a logic symbol without those pins being displayed.

The value of the attribute specifies netname:pinnumber pairs, i.e. “VCC:14,GND:7”.

Caution is advised whenever using hidden pins as a review of the schematic would not permit their visual review.

8.1.5 footprint

The footprint component optional attribute is by the gEDA PCB layout tool as an indicator of which gEDA PCB footprint is associated with a component. If footprints are assigned, the designer should verify that the symbol's pin numbers correctly correspond with the footprint's pin numbers. This is particularly important for items such as 3 pin transistors which have pins assigned in different ways.

8.1.6 pinnumber

The pinnumber pin attribute specifies the pin number associated with a symbol pin. For some tools, such as gEDA and expressPCB, this attribute must be a number.

8.1.7 pinlabel

The pinlabel pin attribute specifies the label associated with a symbol pin. Normally, it would represent the internal signal name (port name) of the part. It is recommended that pinlabel follow VHDL naming conventions, i.e. starting with an alpha character with the rest being alpha, numeric, underscore characters.

8.1.8 pinseq

The pinseq pin attribute is used by gEDA tools. It is a unique sequential number for each pin.

8.1.9 pintype

The pintype pin attribute specifies characteristics about a pin.

Attribute Value	Description	VHDL Mode
in	An input pin	in
out	An output pin	out

Attribute Value	Description	VHDL Mode
io	An inout pin	inout
oc	An open collector/drain output	out
oe	An open emitter output	out
tp	A totem pole output	out
tri	A 3 state output	out
clk	A clock	in
pwr	A power or ground line.	in
pas	Passive (default)	in

8.1.10 description

The description attribute provides a one line description of the device.

8.1.11 author

The optional author attribute provides the author's name and email address.

8.1.12 comment

The comment attribute can contain any other information.

8.1.13 numslots

The numslots attribute indicates the number of possible instances for a symbol which represents one instance of a physical part that contains multiple instances. For example, a resistor pack containing 8 resistors could be represented as a single device containing all resistors or a single resistor with the numslots attribute equal to 8.

8.1.14 slot

The slot attribute specifies which piece of a device is represented by this symbol. The slot can be any integer from 1 to numslots.

8.1.15 slotdef

The slotdef attribute defines the pinnumber assignment for a specific slot. The format is:

slotdef=slot : pinnumber1,pinnumber2,...pinnumberN

where:

slot = the slot number being defined

pinnumbers = comma-separated pinnumber list in the order defined by the pinseq attribute.

Example:

```
numslots=4
slotdef=1:1,2,3
slotdef=2:4,5,6
slotdef=3:10,9,8
slotdef=4:13,12,11
```

8.1.16 documentation

The documentation attribute contains a link for more information about the device and can be a file name or URL.

8.1.17 dist-license

The dist-license attribute specifies the redistribution license for the symbol or schematic.

8.1.18 use-license

The use-license attribute describes the license for how the symbol or schematic is used. Example values are: unlimited, personal use only, not for sale.

8.1.19 symversion

The symversion attribute describes the version of a symbol with the format major.minor.

Example: 1.0

8.2 Special Attributes

Special attributes are defined for use with eightolives' Schematic.

8.2.1 partid

This optional component attribute can be used to specify an inventory part number which may differ from the device attribute. If the partid attribute is specified, it will appear in the BOM listing.

8.2.2 padsdevice

This optional component attribute can be used to specify the part listed in a PADS netlist. If this attribute is not specified, the device attribute will be used for the part in that listing.

8.2.3 sheetnumber

This optional schematic attribute can be used to specify the number of the schematic sheet.

8.2.4 netcomment

This optional net attribute is used to specify the one line comment that appears next to the signal declaration in a VHDL output file. There can only be one non-null netcomment associated with a signal and it appears as an attribute applied to one net segment.

8.2.5 labelissignal

This optional pin attribute is used on “net stub” or “offsheet reference” components. When the attribute exists and has the value “true”, the pinlabel attribute is updated to be that of the connected signal.

8.2.6 ncok

This optional pin attribute is used during check design. If it exists and its value is “true”, a “no connection” will not be listed in the check design output. If it exists and the value is “mandatory”, an error will be listed if there is a connected signal.

8.2.7 model

This optional component attribute is used to specify an associated library. The value “simlib” specifies the eightolives Simulation Library as the source of a functional model for this component. This is saved in VHDL output as a generic and used by the eightolives DesignTool in preparation of circuit simulation.

8.2.8 primitive

This optional component attribute is used to indicate that the symbol represents the lowest level of hierarchy for purposes of simulation. Setting the value to “true” indicates that the symbol represents a simulation primitive. This is saved in VHDL output as a generic and used by the eightolives DesignTool in preparation of circuit simulation.

8.2.9 busname

This optional attribute is used to indicate the name of a bus segment.

8.2.10 size

This optional attribute is used to indicate the number of nets contained in this bus or vector. For vectors, the VHDL equivalent is specifying the vector as std_logic_vector((size -1) downto 0). The size attribute may be set to a component attribute's (generic) name allowing the size to be set by the component attribute. The attribute name should have the prefix “I_” to be recognized as representing a VHDL INTEGER by Schematic Mobile's VHDL generator.

8.2.11 limit1

This optional attribute is used to specify the first number in a vector specification. The VHDL equivalent is specifying the vector as std_logic_vector(limit1 to limit2). If limit1 is greater than limit2, “downto” is implied.

8.2.12 limit2

This optional attribute is used to specify the second number in a vector specification. The VHDL equivalent is specifying the vector as std_logic_vector(limit1 to limit2). If limit1 is greater than limit2, “downto” is implied.

8.2.13 logic_type

This optional attribute is used to specify the VHDL logic type of a net or bus. The default is std_logic for nets and std_logic_vector for vectors.

8.2.14 src

This optional attribute is used to specify the full path to a symbol.

8.2.15 gnd_plane_signal

This optional top level attribute is used to specify the signal name of the ground plane that may be a layer in a printed circuit board artwork using eightolives' Layout tool.

8.2.16 power_plane_signal

This optional top level attribute is used to specify the signal name of the power plane that may be a layer in a printed circuit board artwork using eightolives' Layout tool.

8.2.17 footprint

This optional component attribute is used to specify the name of the footprint file that would be used in a printed circuit board artwork using eightolives' Layout tool.

8.2.18 placenear

This optional component attribute is used to specify the component and pin number near which this component should be placed in a printed circuit board artwork using eightolives' Layout tool.

8.2.19 Timing Attributes

Timing attributes are optional component attributes used to specify timing information for simulation or analysis. A timing attribute name is an upper case “T” followed by lower case letters or digits. Examples are: Td, T4, Tperiod, Thold. The corresponding value must be a time value such as: 3 ns, 120 ps, or 43 us. These are saved in VHDL output as generics of type TIME. They allow timing information to be passed to simulation models.

9 File Formats

The file format for schematic and symbol files is defined by the gEDA project at http://wiki.geda-project.org/geda:file_format_spec. A summary is presented here. Recommendations and notes are included within brackets [].

9.1 Concatenated Schematic Files

For files compatible with gEDA tools, schematic files (.sch) represent one sheet of the schematic. Schematic Mobile also allows all schematic sheet files to be concatenated into a single (.sch) file to simplify design loading. The “version” line (see section 9.3) acts as the “delimiter” between sheets in the concatenated file.

9.2 File names

Schematic sheets are stored with the suffix “.sch” and symbols are stored with the suffix “.sym”. [Meaningful symbol names are recommended. Schematic sheet files can have any name but names such as “Sheet_1” are recommended.]

9.3 Version

The first line of schematic or symbol files shall be the file version indication:
“v 20110115 2”.

9.4 Coordinate Space

The coordinate space defines position (0,0) as the lower left corner of the drawing universe. X coordinates increase to the right. Y coordinates increase upward. Position is represented in thousandths of an inch.[The universe size is 120 inches wide (X max = 120000) and 90 inches high (Y max = 90000).]

9.5 Colors

Colors are specified in drawing objects by a numeric index as defined in Table x.

Index	Color type
0	BACKGROUND_COLOR
1	PIN_COLOR
2	NET_ENDPOINT_COLOR
3	GRAPHIC_COLOR
4	NET_COLOR
5	ATTRIBUTE_COLOR
6	LOGIC_BUBBLE_COLOR
7	DOTS_GRID_COLOR
8	DETACHED_ATTRIBUTE_COLOR
9	TEXT_COLOR
10	BUS_COLOR
11	SELECT_COLOR

Index	Color type
12	BOUNDINGBOX_COLOR
13	ZOOM_BOX_COLOR
14	STROKE_COLOR
15	LOCK_COLOR
16	OUTPUT_BACKGROUND_COLOR
17	FREESTYLE1_COLOR
18	FREESTYLE2_COLOR
19	FREESTYLE3_COLOR
20	FREESTYLE4_COLOR
21	JUNCTION_COLOR
22	MESH_GRID_MAJOR_COLOR
23	MESH_GRID_MINOR_COLOR

[Actual colors associated to each item are defined in the selected color palette. You can edit the color palette from the Preferences Menu. Not all color item are used.]

9.6 Drawing Objects

Drawing Objects are represented by a line consisting of an upper case letter plus item unique parameters separated by a space character. Table x lists the defined objects.

Drawing Object	Type	Used in Schematics	Used in Symbols
Line	L	Y	Y
Box	B	Y	Y
Circle	V	Y	Y
Arc	A	Y	Y
Text and Attributes	T	Y	Y
Path	H	Y	Y
Net	N	Y	N
Pin	P	N	Y
Bus	U	Y	N
Component	C	Y	N
Picture	G	Y	Y

9.7 Line

Position	Field	Type/units	Description
#	type	char	L
1	x1	int / mils	First x coordinate
2	y1	int / mils	First y coordinate
3	x2	int / mils	Second x coordinate
4	y2	int / mils	Second y coordinate
5	color	int	Color index
6	width	int / mils	Width of line
7	capstyle	int	capstyle
8	dashstyle	int	dashstyle
9	dashlength	int / mils	Length of dash
10	dashspace	int / mils	Length of space

Capstyle: END_NONE = 0, END_SQUARE = 1, END_ROUND = 2.

Dashstyle: TYPE_SOLID = 0, TYPE_DOTTED = 1, TYPE_DASHED = 2, TYPE_CENTER = 3, TYPE_PHANTOM = 4.

9.8 Box

Position	Field	Type/units	Description
#	type	char	B
1	x1	int / mils	Lower left x coordinate
2	y1	int / mils	Lower left y coordinate
3	width	int / mils	Width of box
4	height	int / mils	Height of box
5	color	int	Color index
6	width	int / mils	Width of line
7	capstyle	int	capstyle
8	dashstyle	int	dashstyle
9	dashlength	int / mils	Length of dash
10	dashspace	int / mils	Length of space
11	filltype	int	
12	fillwidth	int / mils	
13	angle1	int / degrees	

Position	Field	Type/units	Description
14	pitch1	int / mils	
15	angle2	int /degrees	
16	pitch2	int / mils	

Capstyle: END_NONE = 0, END_SQUARE = 1, END_ROUND = 2.

Dashstyle: TYPE_SOLID = 0, TYPE_DOTTED = 1, TYPE_DASHED = 2, TYPE_CENTER = 3, TYPE_PHANTOM = 4.

Filltype: FILLING_HOLLOW = 0, FILLING_FILL = 1, FILLING_MESH = 2, FILLING_HATCH = 3,

9.9 Circle

Position	Field	Type/units	Description
#	type	char	V
1	x1	int / mils	Center x coordinate
2	y1	int / mils	Center y coordinate
3	radius	int / mils	Radius of circle
4	color	int	Color index
5	width	int / mils	Width of line
6	capstyle	int	capstyle
7	dashstyle	int	dashstyle
8	dashlength	int / mils	Length of dash
9	dashspace	int / mils	Length of space
10	filltype	int	
11	fillwidth	int / mils	
12	angle1	int /degrees	
13	pitch1	int / mils	
14	angle2	int /degrees	
15	pitch2	int / mils	

Capstyle: END_NONE = 0, END_SQUARE = 1, END_ROUND = 2.

Dashstyle: TYPE_SOLID = 0, TYPE_DOTTED = 1, TYPE_DASHED = 2, TYPE_CENTER = 3, TYPE_PHANTOM = 4.

Filltype: FILLING_HOLLOW = 0, FILLING_FILL = 1, FILLING_MESH = 2, FILLING_HATCH = 3,

9.10 Arc

Position	Field	Type/units	Description
#	type	char	A
1	x1	int / mils	Center x coordinate
2	y1	int / mils	Center y coordinate
3	radius	int / mils	Radius of arc
4	startangle	int / degrees	Starting angle of the arc
5	sweepangle	int / degrees	Amount the arc sweeps
6	color	int	Color index
7	width	int / mils	Width of line
8	capstyle	int	capstyle
9	dashstyle	int	dashstyle
10	dashlength	int / mils	Length of dash
11	dashspace	int / mils	Length of space

Capstyle: END_NONE = 0, END_SQUARE = 1, END_ROUND = 2.

Dashstyle: TYPE_SOLID = 0, TYPE_DOTTED = 1, TYPE_DASHED = 2, TYPE_CENTER = 3, TYPE_PHANTOM = 4.

9.11 Text and Attributes

Position	Field	Type/units	Description
#	type	char	T
1	x1	int / mils	First x coordinate
2	y1	int / mils	First y coordinate
3	color	int	Color index
4	size	int / points	Font size
5	visibility	int	0 = invisible, 1 = visible
6	show_name_value	int	0 = both, 1 = value, 2 = name
7	angle	int / degrees	Angle of text
8	alignment	int	Alignment / origin of text
9	num_lines	int	Number of lines
10	String lines 1 .. n	string	Text strings on separate lines

An attribute differs from text in that there is only 1 line with the format:

attribute_name=attribute_value

Alignment is defined by an integer referring to the origin location as shown in Figure 9-1.

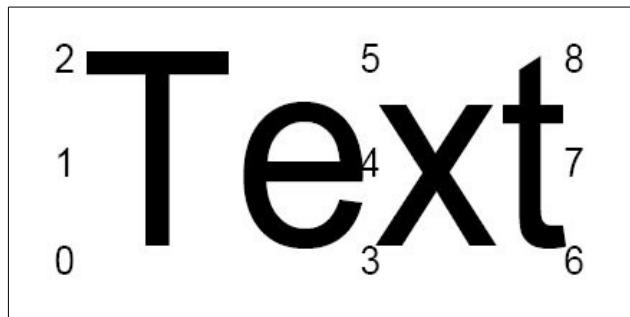


Figure 9-1 Text Alignment

9.12 Net

Position	Field	Type/units	Description
#	type	char	N
1	x1	int / mils	First x coordinate
2	y1	int / mils	First y coordinate
3	x2	int / mils	Second x coordinate
4	y2	int / mils	Second y coordinate
5	color	int	Color index

9.13 Pin

Position	Field	Type/units	Description
#	type	char	P
1	x1	int / mils	First x coordinate
2	y1	int / mils	First y coordinate
3	x2	int / mils	Second x coordinate
4	y2	int / mils	Second y coordinate
5	color	int	Color index
6	pintype	int	0 = normal pin, 1 = bus pin

Position	Field	Type/units	Description
7	whichend	int	0 = (x1,y1), 1 = (x2,y2)

9.14 Component

Position	Field	Type/units	Description
#	type	char	C
1	x	int / mils	origin x coordinate
2	y	int / mils	origin y coordinate
3	selectable	int	0 = not selectable, 1 = selectable
4	angle	int / degrees	0, 90, 180 or 270
5	mirror	int	0 = normal, 1 = mirrored
6	basename	string	Filename of symbol

9.15 Bus

[In eightolives' Schematic, Bus is only implemented as a graphic element.]

Position	Field	Type/units	Description
#	type	char	U
1	x1	int / mils	First x coordinate
2	y1	int / mils	First y coordinate
3	x2	int / mils	Second x coordinate
4	y2	int / mils	Second y coordinate
5	color	int	Color index
6	ripperdir	int	0 = no ripper, 1 , -1 (Note: Schematic Mobile does not implement rippers.)

9.16 Picture

[Encoded picture is not supported by eightolives' Schematic.]

Position	Field	Type/units	Description
#	type	char	G
1	x1	int / mils	Lower left x coordinate
2	y1	int / mils	Lower left y coordinate
3	width	int / mils	Width of picture
4	height	int / mils	Height of picture
5	angle	int / degrees	Angle 0, 90, 180, 270

Position	Field	Type/units	Description
6	mirrored	int	0 = normal, 1 = mirrored
7	embedded	int	0 = not embedded
8	filename	string	Filename of picture
9	Encoded data	string	Serialized picture data base64
10	Encoded data end	string	A line containing only a dot

9.17 Path

Position	Field	Type/units	Description
#	type	char	H
1	color	int	Color index
2	width	int / mils	Width of line
3	capstyle	int	capstyle
4	dashstyle	int	dashstyle
5	dashlength	int / mils	Length of dash
6	dashspace	int / mils	Length of space
7	filltype	int	
8	fillwidth	int / mils	
9	angle1	int / degrees	
10	pitch1	int / mils	
11	angle2	int / degrees	
12	pitch2	int / mils	
13	num_lines	int	
14	path_data 1 - n	path data	Path data is on separate lines.

Capstyle: END_NONE = 0, END_SQUARE = 1, END_ROUND = 2.

Dashstyle: TYPE_SOLID = 0, TYPE_DOTTED = 1, TYPE_DASHED = 2, TYPE_CENTER = 3, TYPE_PHANTOM = 4.

Filltype: FILLING_HOLLOW = 0, FILLING_FILL = 1, FILLING_MESH = 2, FILLING_HATCH = 3,

9.17.1 Path Data

Command	Name	Parameters	Description
M	moveto	x, y	
L	lineto	x, y	

C	curveto	x1, y1, x2, y2, x, y	Not implemented
Z or z	closepath		

10 Compatibility Exceptions to gEDA

Eightolives' Schematic Mobile has the following compatibility exceptions to gEDA items.

11 Services

A service represents an entity that is used by Schematic Mobile for file storage.

11.1 Local Service

Local service is a web browser's cache memory. It provides storage of .sch and .sym files on the computing device being used.

The disadvantages of Local storage include the risk of all file loss if cache memory is cleared either intentionally by the user or because of a resetting browser condition. Backing-up files to more permanent storage is recommended. The size of cache available may also be limited by the computing device's memory or software settings.

The advantage of Local storage is that Schematic Mobile can interface with it seamlessly and projects can be saved and retrieved automatically.

11.2 Server Service

Server service is the file services provided by the server from which you have executed Schematic Mobile.

When you start Schematic Mobile from the eightolives web site, the eightolives server provides read-only access to example projects and libraries.

If you have installed Schematic Mobile as a tool in eightolives Microserver on a PC, the Microserver will provide read and write capabilities to the PC's projects directories providing a more permanent storage of schematic and symbol files. Microserver can also provide remote access to other symbol libraries installed on that PC.

If you have installed Schematic Mobile as a tool on some other server, you can access file services only if the server implements the Schematic Mobile Server Interface API.

11.3 External Service

External service allows executing Schematic Mobile from the eightolives web site while using file services of an external Microserver running on a PC on the local network or an external cloud service.

When a Microserver is used as an External Service, the Microserver will provide read and write capabilities to the PC's projects directories providing a more permanent storage of schematic and symbol files. Microserver can also provide remote access to other symbol libraries installed on that PC.

Schematic Mobile's External Services popup window includes a discovery tool which will scan your local network for active Microservers simplifying access to that resource.

Other servers can be used for External service if they implement the Schematic Mobile Server Interface API.

Access to commercial "cloud" services require specific driver code for each service to correctly login and access files. No drivers are currently available. Contact eightolives for requests or more information.

11.4 Drive Service

Drive service lets you store and retrieve files from your Google Drive account. Upon selecting Drive, Google interface software will then be loaded and executed. If you are not logged in to Google Drive, the Google Login Window will be displayed.

When successfully logged in, the words “Google Drive” will appear as the top level project option. Folders at the top level are interpreted as “projects” which can be individually selected.

Note: If you select Drive, you are subject to Google's terms and conditions.

11.5 A Computer's File System

Although not a service as defined herein, the computer running Schematic Mobile may have an accessible, traditional file system where schematic and symbol files may be stored. Some computing devices such as Apples iPod do not provide this capability.

The disadvantage of using the file system is that interaction with Schematic Mobile via the web browser is not automatic. Each schematic sheet and symbol file must be saved and retrieved manually making the process time consuming and error-prone.

The advantage of file system access is reliable storage of the data and the ability to directly access other libraries installed on the computer.

Access to the file system is independent of the service that is being used.

12 Schematic Mobile Server Service Interface

This section describes the interface to the Server Service. It is implemented by the eightolives web site for read-only operations.

12.1 Base URL

The Base URL is the URL that specifies the directory that contains the file “projects.rss”. This file contains descriptions and links to projects available on the server. The directory also contains the file “libraries.rss” which contains descriptions and links to default libraries supported by the server.

12.1.1 projects.rss

The projects.rss file contains links to a project directory where schematic and symbol files are kept. An example projects.rss file is as follows:

```
<?xml version="1.0"?>

<rss version="2.0">
<channel>
<item>
<title>Demo</title>
<link>demo/upload</link>
<description>Demo project.</description>
</item>
</channel>
</rss>
```

If the link is or contains “PROJECT_LIB”, the actual project path will replace “PROJECT_LIB”. If the starts with “http”, the specified URL will be used. If the URL is a relative expression, it will be relative to the baseURL directory.

12.1.2 Project Directory

The URL of the project directory is specified in the projects.rss file. The project directory contains a file “libraries.rss” which contains descriptions and links to libraries supported by the project. The links in “libraries.rss” point to rss files that contain descriptions and links to actual .sym and .sch symbol files or to directories that will respond with a list of html links to files.

The project directory also contains the file “files.rss” which lists files in the project directory.

13 Feedback

Feedback for this document or aspects of the tool can be sent by email to feedback@eightolives.com