

William_Kaupinis@eightolives.com June 1, 2014

eightolives.com

 Eightolives' Schematic is a graphical design entry tool used to capture an electronic circuit idea.

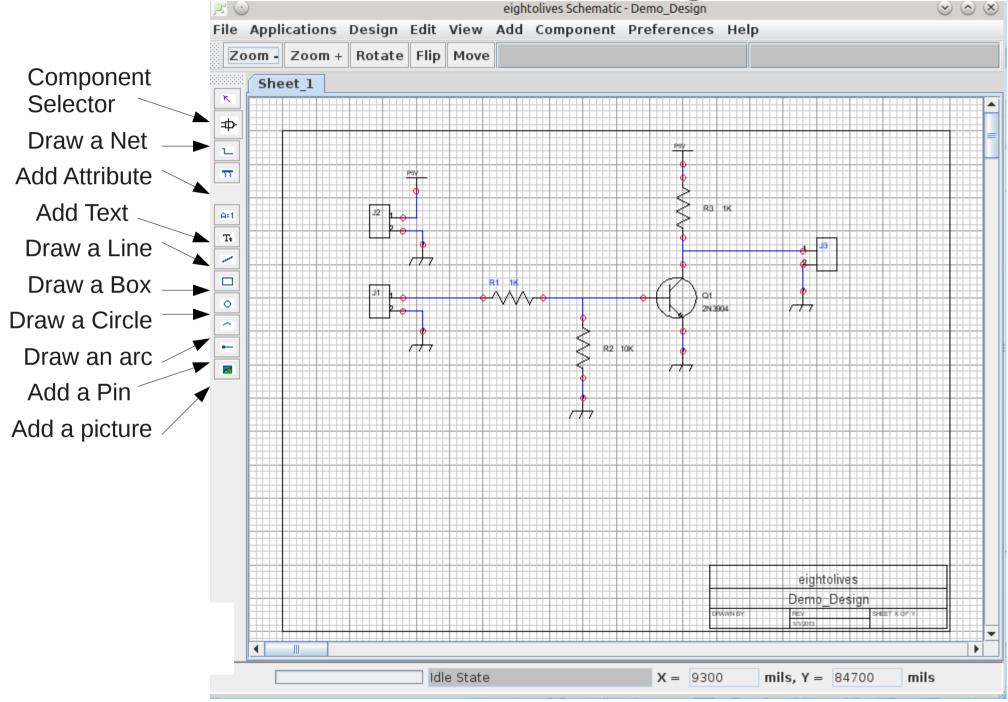
With a schematic you can create a netlist for fabricating a printed circuit board, generate a VHDL file for simulation, or just prepare a graphic representation of a design.

- You place symbols that represent parts and interconnect their pins with nets
- You run DesignCheck and then output a netlist for circuit board design or VHDL for simulation

eightolives.com Getting Started..

- Install and start eightolives' Schematic (see end of presentation)
- Menu option File > New > Design Project creates a design directory for your work
- Clicking the Component Selector button opens the Component Selector window where you can double click parts, drag and click to place, right click for options
- Click the Net button to draw wires to connect pins
- Menu option *Design > Check Design* to itemize errors
- Menu option Design > Generate Netlist > gEDA PCB Format to output a netlist for the gEDA PCB artwork layout tool or choose PADS or expressPCB or EESchema (KiCad) format
- Menu option *File > Save All* to save your design

eightolives.com



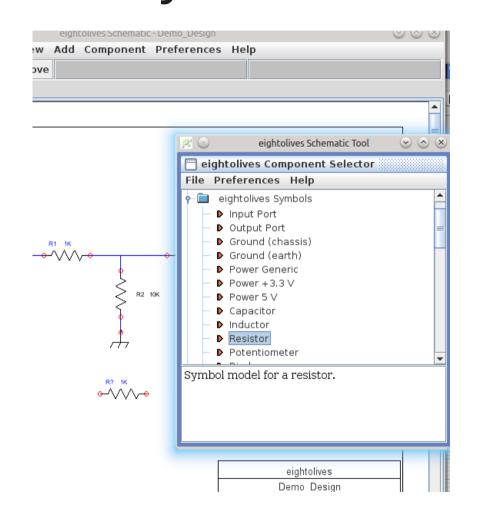
eightolives.com Basic Definitions

- A schematic *project* is a defined directory structure that contains files for each schematic sheet (.sch file) and files for each component symbol (.sym file)
- A schematic *sheet* is a file that contains graphics, attributes, text, component symbols and nets.
- A *symbol* contains graphics, attributes, text and pins that represent the part's inputs and outputs.
- A *net* represents a wire segment used to interconnect pins of the various parts.
- An *attribute* is a is name/value string pair used to customize information. Many attributes are predefined for various purposes.

eightolives.com Project directory structure

```
Project_name
     schematic
          Project name Sheet 1.sch
          Project name Sheet 2.sch
          symbols
               RESISTOR 1K.sym
               CAPACITOR 10uF.sym
               SN74HCT00.sym
               RESISTOR_1K.sym
```

eightolives.com Use symbols from an existing library



Copyright $\ensuremath{\mathbb{C}}$ 2014 William Kaupinis All Rights Reserved

- The Component Selector window lets you select basic, generic symbols from the on-line eightolives' symbol library
- Double click an item from the selector, drag it on the schematic and click to place
- You can also use symbol libraries from the Gnu gEDA project or KiCad if you have them installed. Component Selector can list items in those libraries too.

eightolives.com Your link to symbols

<u>R</u> 🔾	eightolives Schematic Tool 🛛 😒 🖄
🦳 ei	ghtolives Component Selector
File	Preferences Help
▶ Sc	hematic Tool 🔶
- 🖻	Resources
• 🛅	eightolives Symbols
- 🖻	eightolives Parts
e- 🛅	gEDA Symbols
e 💼	KiCad Symbols
	T4xgxx.dcm
<u>م</u>	🛅 74xx.dcm
	- \$ <u>+</u> 7400
	- ⁴ ⊭ 74AHC1G14 ▼
	IC1G14 ting buffer with Schmitt trigger.
mver	thig barrer with schning digger.

- The Component Selector Window also has a Resources folder with links to documentation, help and examples.
- A Personal Bookmarks folder stores links to design files, web sites or documentation that you can specify.

eightolives.com Or use the Built-In Wizards

- Block Symbol Wizard makes block (black box) symbols with pins possible on all 4 sides. Use this for high pin-count parts or for symbols that "look like" the part (DIPs, SOIC, QFP) or for any type of part.
- Connector Wizard makes generic connectors.
- Header Wizard makes Dual-row Header symbols.
- Logic Symbol Wizard makes and, nand, or, nor, xor and xnor gates.

eightolives.com **Block Symbol Wizard**

eightolives Schematic - Demo_Design 📀	
ications Design Edit View Add Component Preferences Help	TS3A5017 07 00 00 00 00 00 00 00 00 00 00 00 00
	S12 GND RA
Zoom + Rotate Flip Move	
et_1	510 77.7
🗾 🛒 🕗 🛛 🛛 Block Symbol Wizard 🛛 😒 🔿 🛞	Edit Connections
Reference Designator U?	Edit Attributes
	Assign Footprint
	1 Toggle Pin Numbers
Device Name DIP14	Move
	Flip
Number of Pins on Left Side 7 Start Pin 1	- un Rotate
	L si3 Change Symbol Name
	G Save Symbol
Number of Pins on Right Side 7	Сору
	LEN2N View Source
Number of Pins on Top 0	About
	2 S22 Delete
	ulszi
Number of Pins on Bottom 0	
	Right clicking a symbol
Minium Width (in thousanths) 1500	
	gives you a popup
Minium Height (in thousanths) 500 📃 Half size	menu.
Make Symbol Cancel	Select Edit
	Connections to change
	pin names, or connect
Copyright © 2014 William Kaupinis All Rights Reserved	nets.

13

eightolives.com Edit Connections Window

0			Edit	Connections	D	\odot	0 8
Net Name	Pin Number	Pin Label	Туре	Pintype	Comment		
	1	P1	inout	io			
	10	P10	inout	io			
	11	P11	inout	io			
	12	P12	inout	io			
	13	P13	inout	io			
	14	P14	inout	io			
	2	P2	inout	io			
	3	P3	inout	io			
	4	P4	inout	io			
	5	P5	inout	io			
	6	P6	inout	io			
	/		inout				
	9	P9	inout	io			
	7 8 9	P7 P8 P9	inout	io io			
		Delete Rov	ws	Update Ports	Cancel		

Use this window to edit pin labels, pin numbers or connected net names.

Select a symbol, right click and select the Edit Connections option.

Copyright © 2014 William Kaupinis All Rights Reserved

eightolives.com Edit Attributes Window

The Edit Attributes popup option lets you change an attribute value, add a new attribute or edit details of an attribute's characteristics.

I	2 🖸	Component Edi	tor - DCompone	ent	000	×
	Operations He	lp				
	Attribu	teName	At	ttribute∨alue		•
	refdes		U?			
	device		DIP14			
						=
14						
13						
12						
10						
P9.						
P/B						
E						
						•
	Add New Entr	y Edit Select	ed Entry	Commit	Cancel	
			1P	enip_Design		

eightolives.com Edit Selected Entry Allows Additional Attribute Editing

🛒 🖸	Edit Entry	\odot \otimes \otimes
	Attribute Name: refdes	
	Attribute Value: U?	
	Fort size 10	2
	Font size 10	
	✓ Visible	
	Show Attribute Name	
	Show Attribute Value	
	Angle 0 🔽	
	Alignment 0 🗸	
	Color Index ATTRIBUTE_COLOR	
	Commit Cancel	
	Add New Entry Edit Selected Entry Cor	nmit Cancel
	Uniter and the second sec	uesign

Copyright © 2014 William Kaupinis All Rights Reserved

eightolives.com Some key attributes

• For symbols:

- refdes uniquely identifies an instantiated part
 - U1 (integrated circuit), R3 (resistor), C5 (capacitor)
- device uniquely identifies a symbol for a part
 - RESISTOR_1K, CONNECTOR_D9, CAPACITOR_1uF
- footprint name of the artwork symbol used by a gEDA PCB
- net defines hidden pins sometimes used for power and ground connections

• For nets:

- netname the name of the net
- For pins:
 - pinnumber the package pin number
 - pinlabel the package name for that pin
 - pinseq an incrementing number reference for that pin
 - pintype describes pin function as in, out, io, oe, oc, tp, clk, pwr or pas

eightolives.com Special attributes

- **primitive** a value of "true" indicates that the symbol represents the lowest hierarchy level for simulation
- model a value of "simlib" indicates that the original simulation model can be retrieved from the eightolives on-line simulation library
- Td an upper case "T" followed by other charaters is interpreted at as timing parameter such as Tdelay or Tperiod. Values should be a constant time value such as 10 ns

eightolives.com You can make your own symbols

- Draw a symbol using lines, circles, boxes, text, attributes and pins.
- Group them to make the symbol
- Edit Connections to change pin labels
- Edit Attributes to modify attributes
- Save the symbol.

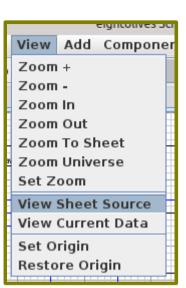
eightolives.com

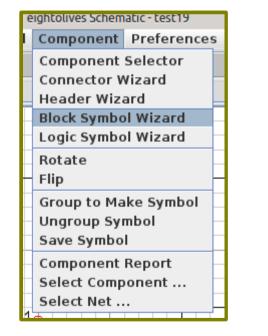
20		
File	Applications Des	sign Edit View /
Оре	n Project	tate Flin Move
New	r 🕨 🕨	Design Project
Оре	n File	Sheet
Open Project Symbol		Empty Sheet
Save	e Sheet As	
Save	e All	
Print	t	P3V
Ехро	ort image	•••••
Clos	e	2 1
Exit		

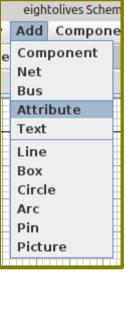
Edit	View	Add	Compon
Undo Redo	-		
	butes et Attri	butes	
Fix U	nassig	ined R	lefdes
Dele	te		Q

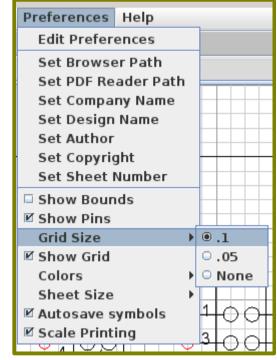
The Menus

Design Edit View	Add Component Pre
New Design Add Sheet	e
Update Nets Check Design	
Generate Netlist →	gEDA PCB Format
Generate VHDL Generate BOM	expressPCB Format PADS Format
Delete Sheet	EESchema Format









0)		
•	Applications	Design	Ec
Zc	Command Processor		F
8	Collect Garbage		
	Memory Status		

Help
Help
License
Get User Manual
Check for Updates
About eightolives' Schematic

Copyright $\ensuremath{\mathbb{C}}$ 2014 William Kaupinis All Rights Reserved

eightolives.com You can write and execute your own scripts or access scripts from eightolives' on-line library

- Create custom checking, editing and netlist tools using ECMA Javascript
- Use scripts to globally modify or add attributes
- Create custom design reports
- Basic Javascript command format is
 - object.method(arg1, ...);

eightolives.com Execute Javascript Scripts Using the Command Processor

🏸 💿 Select a File 😒 🛇 😣	🖉 🕑 Command Processor 🕑 📀 🛞	r
	File Ops Templates Preferences	
List componments, connections Highlight unconnected nets Report Footprints	//Welcome to the Javascript Command Processor Frame //predefined variables are: // frame - The parent frame window // cpf - this Command Processor Frame // cp - the Command Processor // tabbedPane - the JTabbedPane object	111
Lists the components, pins and assigned footprint.	// ste - the SchematicToolEnvionment object	***
Select Cancel		
	comp = sh.getComponentByRefdes("U1");	j,
		1111

The Command Processor executes ECMA Javascript commands from the lower command line, scripts from saved files or scripts from the on-line library. Predefined object variables and often used command templates make programmatic interaction easy.

Copyright $\ensuremath{\mathbb{C}}$ 2014 William Kaupinis All Rights Reserved

eightolives.com Templates for often used commands

1	Command Processor Solo Sing
	Templates Preferences
t	cpf.print();
	cpf.listObjectMethods(cpf);
-	i = tabbedPane.getSelectedIndex();
h	sif = getSchematicInternalFrame(i);
e	sh = sif.getSchematicSheet();
d	sheets = sh.getSchematicSheets();
12	vdojs = sh.getDrawingObjects();
e	vcomps = sh.getLocalDComponents();
e	vpins = comp.getPins();
I	vnets = sh.getLocalDNets();
I	it = vnets.iterator();
	while(it.hasNext()) {net = it.next(); netname = net.getAttributeValue("netname"); cpf.print(netname);}
	comp = sh.getComponentByRefdes("U1");
	pin = comp.getPinByPinnumber("1");

Use the **cpf.listObjectMethods(objectname)**; command to see what methods are available for an object.

Copyright © 2014 William Kaupinis All Rights Reserved

eightolives.com Getting Schematic

- You subscribe to Schematic from the eightolives' web site and receive a user name and password
 - A subscription gets you software updates for 1 year.
- Log in at the eightolives User Login page, select Schematic and download to your Downloads directory EightolivesInstall.exe (Windows) or EightolivesInstall.tar.gz (Linux)
- For Windows: execute EightolivesInstall
- For Linux: tar -xvzf EightolivesInstall.tar.gz Then execute EightolivesInstall

eightolives.com Schematic is installed in your Home Directory

 EightolivesInstall creates a directory structure in your Home Directory and installs all software there

your_home_directory

.eightolives

Schematic

Schematic.cmd (windows), Schematic (Linux)

 Schematic is started by executing Schematic in the .eightolives\Schematic subdirectory

eightolives.com Prerequisites

- Schematic is a Java application and requires Oracle's Java to be installed on your computer. You normally install Java from http://www.java.com
- It is useful to define the environment variable JAVA_HOME when you have multiple versions of Java installed

eightolives.com Software Integrity

- The EightolivesInstall downloads are digitally signed. The digital signatures can also be downloaded when you download the software.
- The eightolives Public Key is downloadable from http://www.eightolives.com/eightolives_key.gpg
- If you use eightolives' Go, the eightolives application launcher, with Gnu gpg installed, Go will automatically verify signatures for all eightolives downloads
- Go is also available on CD which avoids use of the Internet during install

eightolives.com

- Can I use gEDA library symbols? Yes.
- Will gEDA tools work with eightolives schematics, symbols and netlists? Yes. (Follow gEDA setup instructions to add link to your project's symbol library)
- Can I open and work with a gEDA schematic? Yes, if you have the appropriate symbol libraries installed.
- Can I use EESchema (KiCad) symbol libraries? Yes.
- Can I zip my project directory and share my design and symbols? Yes.
- Can I generate other netlist formats? Built-in netlist formats are for gEDA PCB, PADS, expressPCB and EESchema (KiCad). For other formats, you can either use the gEDA tool gsch2pcb tool which supports many formats, create your own netlist program by modifying an available Javascript template, or check for additional netlist options in the eightolives script library.

eightolives.com For more information

- Visit http://www.eightolives.com
- For the latest Java download, visit http://www.java.com
- For more information on GnuPrivacyGuard (gpg) visit http://www.gnupg.org
- For GnuPrivacyGuard downloads visit http://www.gnupg/download/
- For the gEDA Project Wiki visit http://wiki.geda-project.org