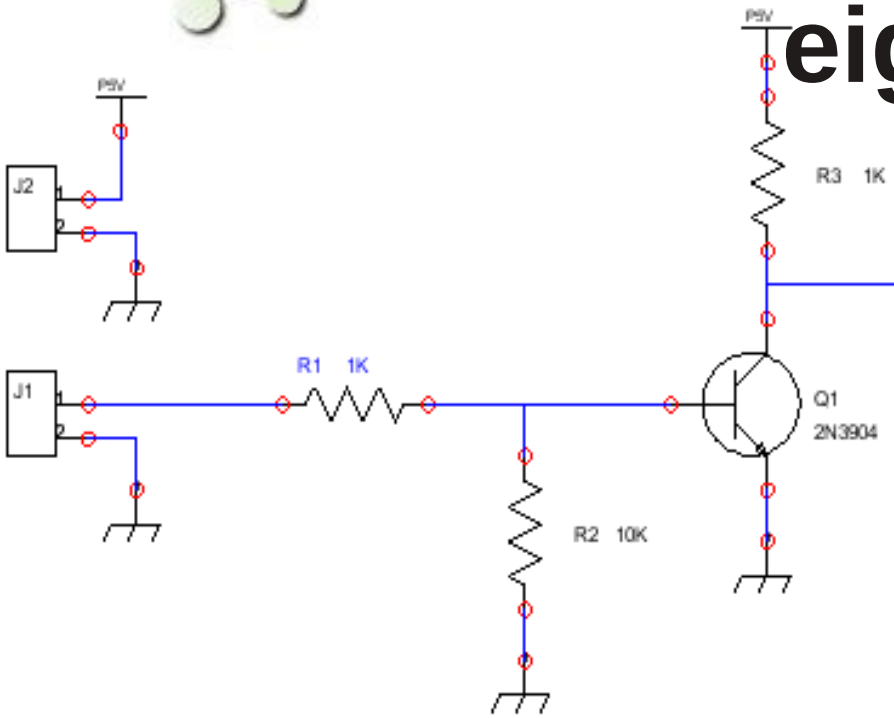
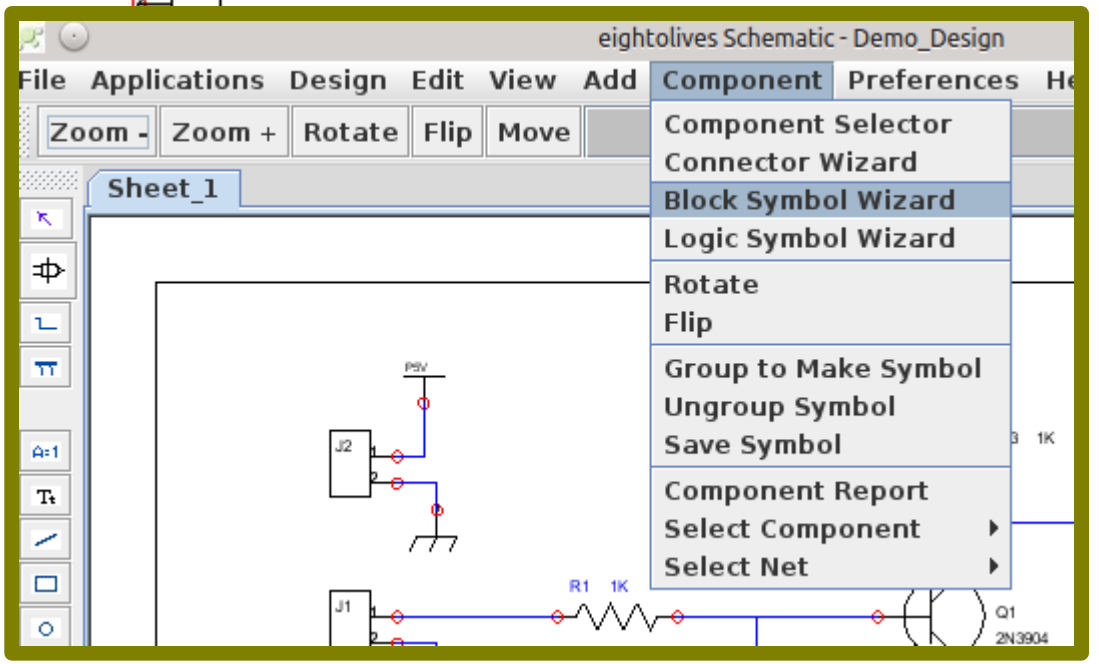
Using eightolives' Schematic





William_Kaupinis@eightolives.com
June 1, 2014

Copyright © 2014 William Kaupinis All Rights Reserved

- 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

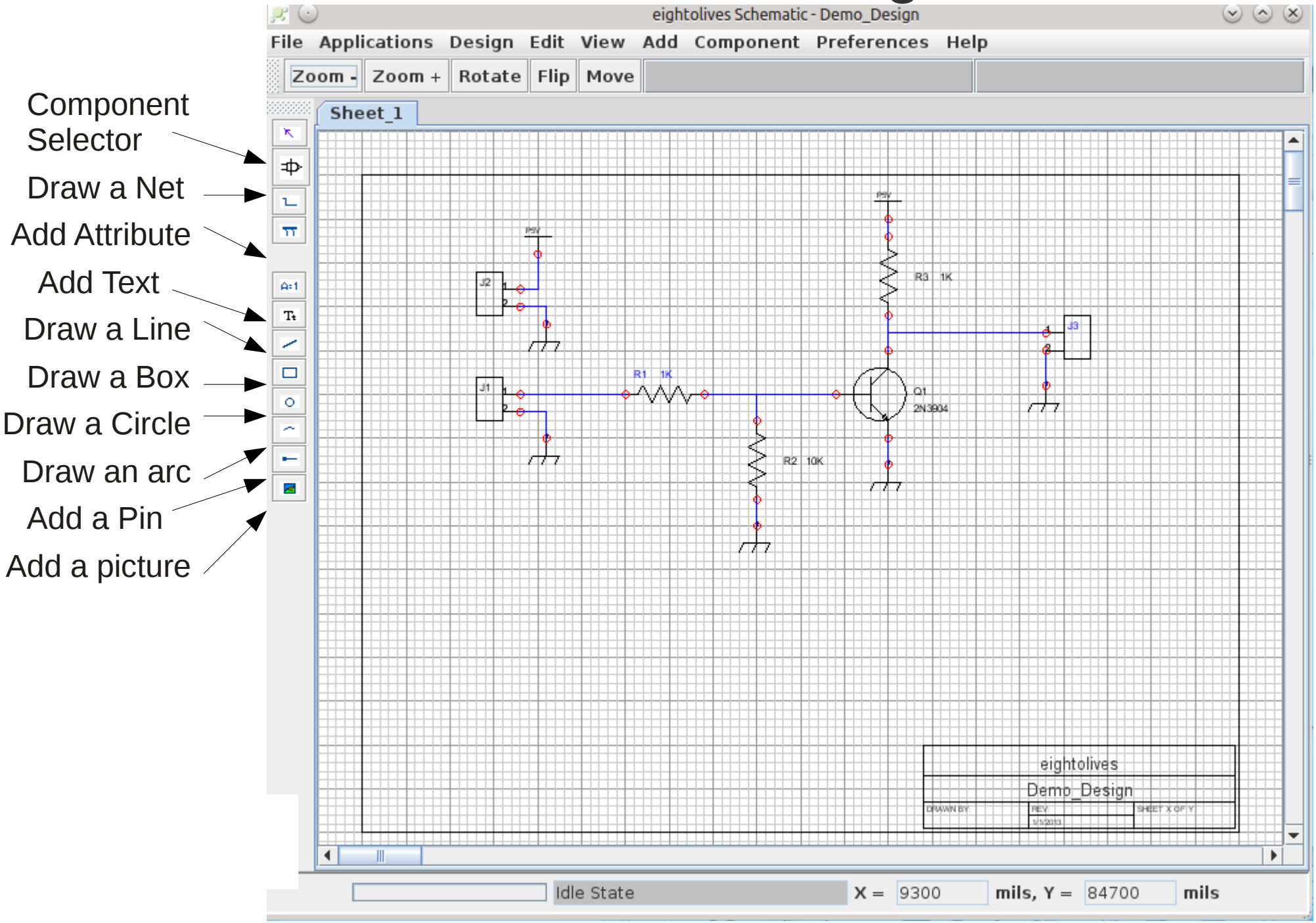
Copyright © 2014 William Kaupinis All Rights Reserved

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

Copyright © 2014 William Kaupinis All Rights Reserved

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.

Copyright © 2014 William Kaupinis All Rights Reserved

Project directory structure

Project_name

schematic

Project_name_Sheet_1.sch

Project_name_Sheet_2.sch

symbols

RESISTOR_1K.sym

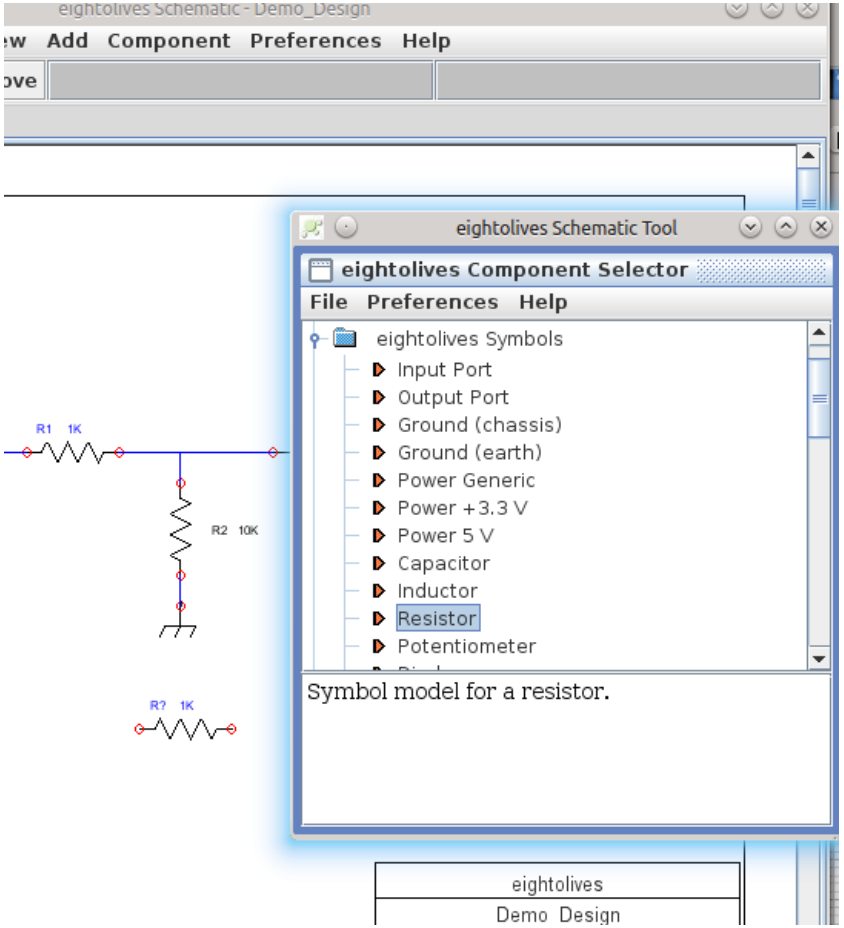
CAPACITOR_10uF.sym

SN74HCT00.sym

RESISTOR_1K.sym

Copyright © 2014 William Kaupinis All Rights Reserved

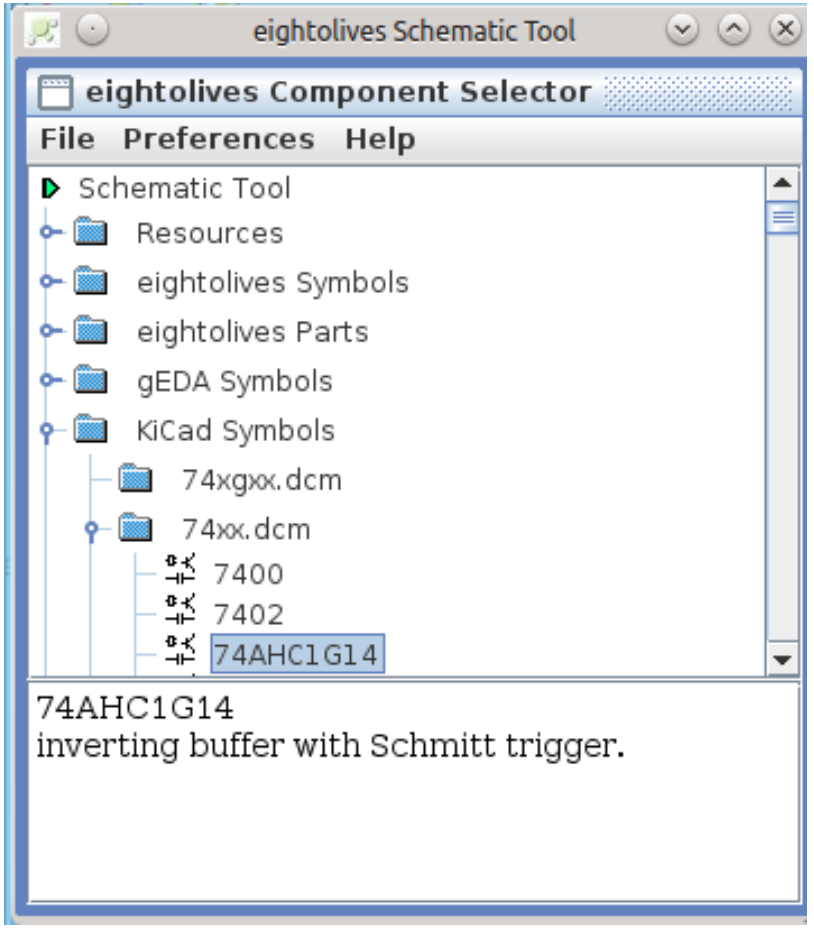
Use symbols from an existing library



- 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.

Copyright © 2014 William Kaupinis All Rights Reserved

Your link to symbols



- 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.

Copyright © 2014 William Kaupinis All Rights Reserved

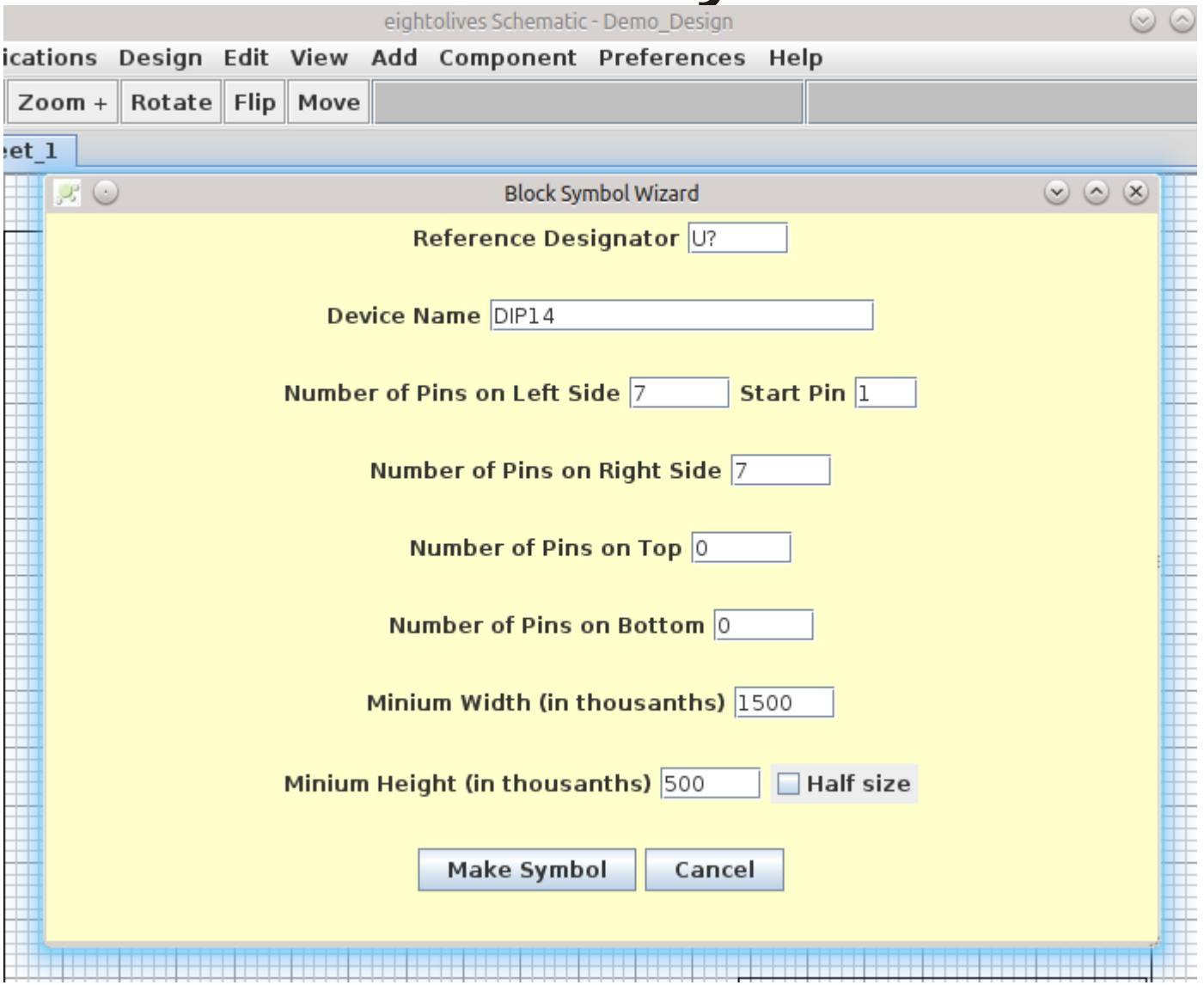
Or use the Built-In Wizards

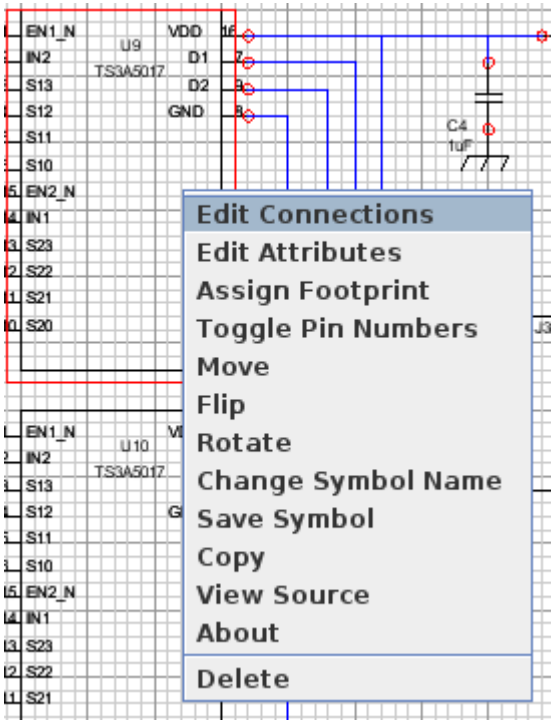
- Block Symbol Wizard makes block (black box) symbols with pins possible on all 4 sides. Use this for high pin-count parts or for symbols that “look like” the part (DIPs, SOIC, QFP) or for any type of part.
- Connector Wizard makes generic connectors.
- Header Wizard makes Dual-row Header symbols.
-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

Use Attribute values to specify refdes, device.

Copyright © 2014 William Kaupinis All Rights Reserved

Block Symbol Wizard



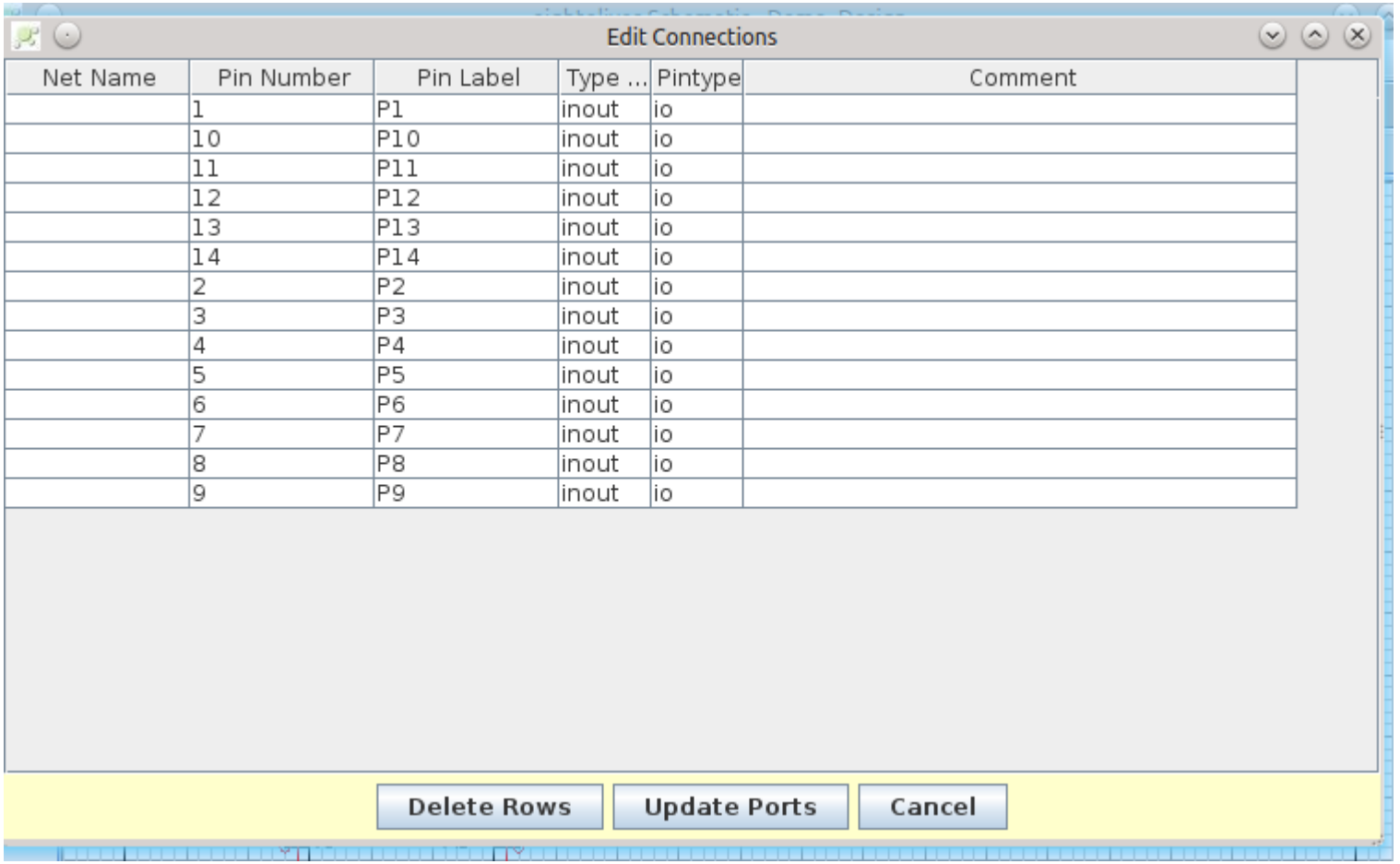


Right clicking a symbol gives you a popup menu.

Select Edit Connections to change pin names, or connect nets.

Copyright © 2014 William Kaupinis All Rights Reserved

Edit Connections Window



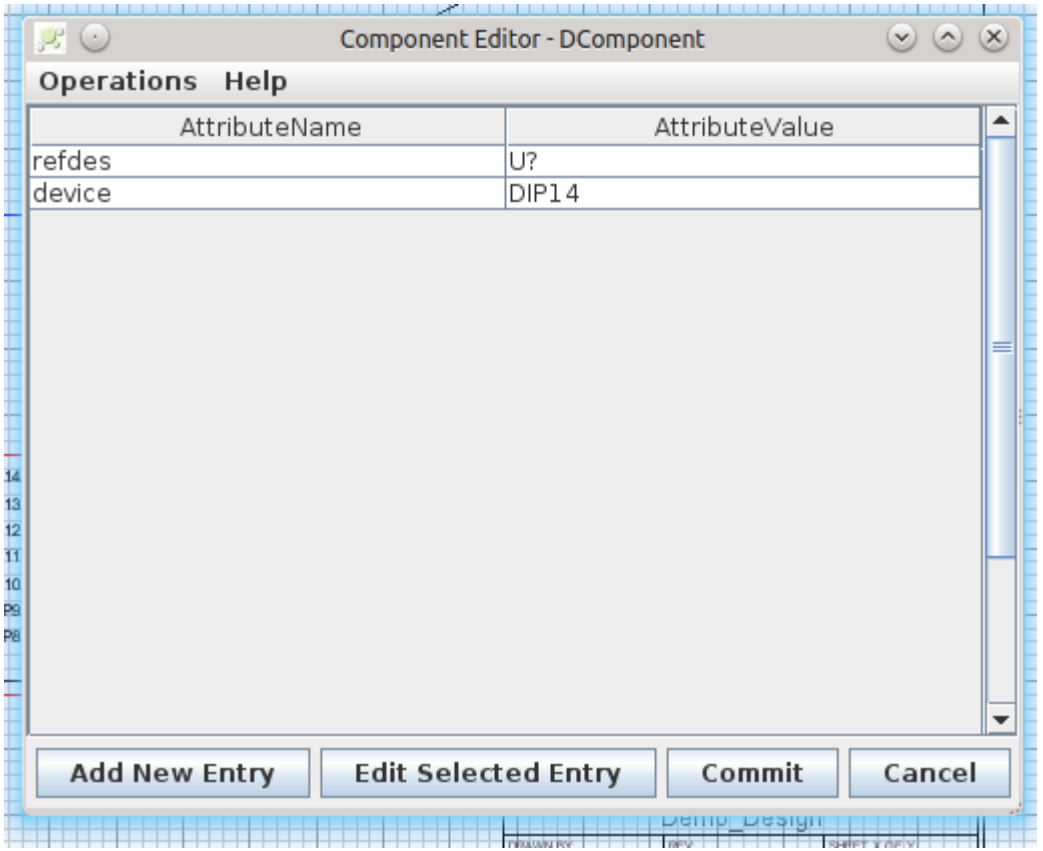
Use this window to edit pin labels, pin numbers or connected net names.

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

Copyright © 2014 William Kaupinis All Rights Reserved

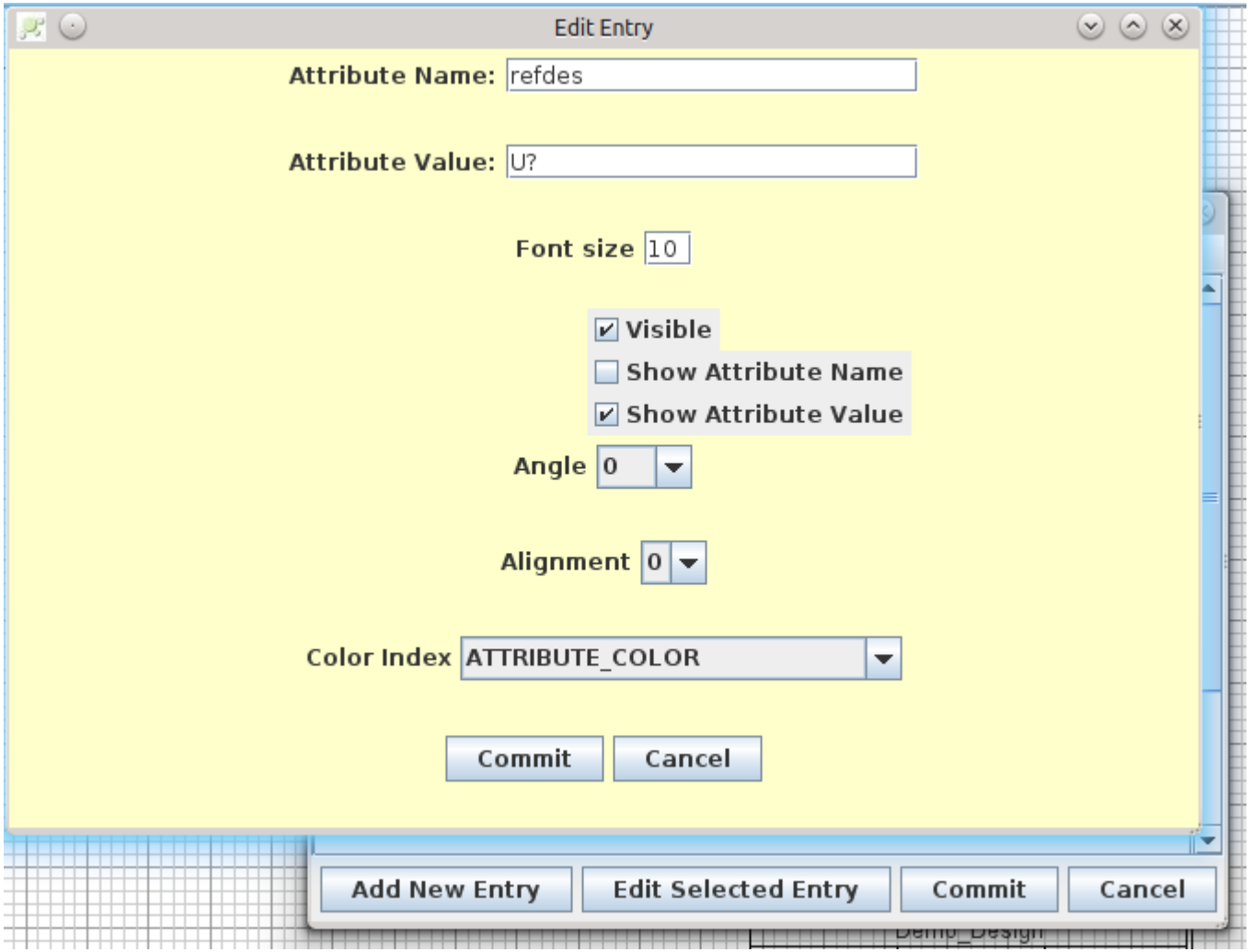
Edit Attributes Window

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



Copyright © 2014 William Kaupinis All Rights Reserved

Edit Selected Entry Allows Additional Attribute Editing



Copyright © 2014 William Kaupinis All Rights Reserved

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

Copyright © 2014 William Kaupinis All Rights Reserved

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

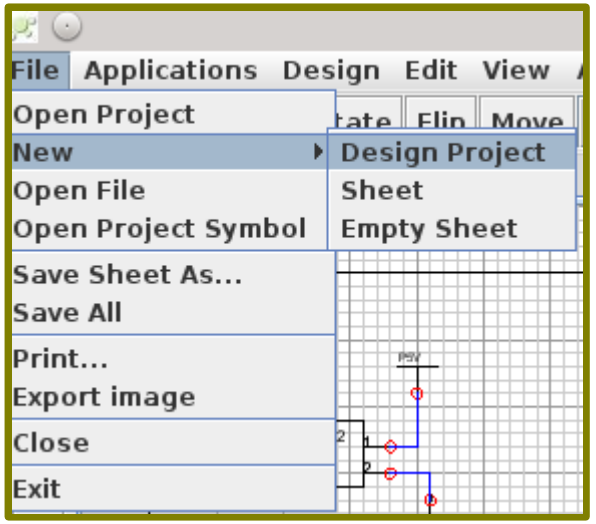
Copyright © 2014 William Kaupinis All Rights Reserved

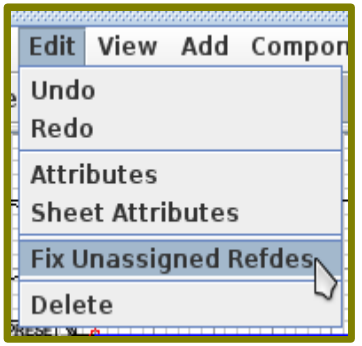
You can make your own symbols

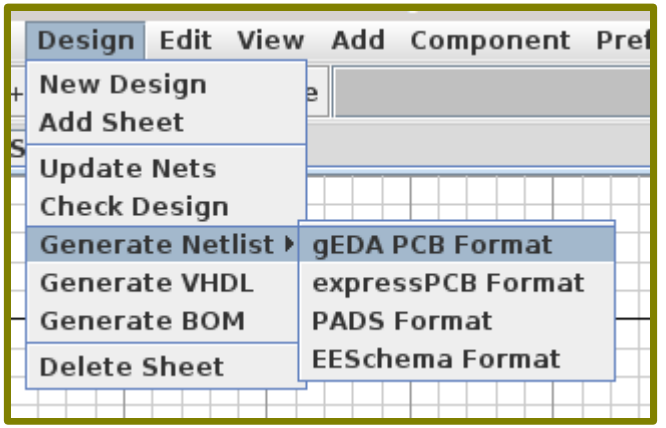
- Draw a symbol using lines, circles, boxes, text, attributes and pins.
- Group them to make the symbol
- Edit Connections to change pin labels
- Edit Attributes to modify attributes
- Save the symbol.

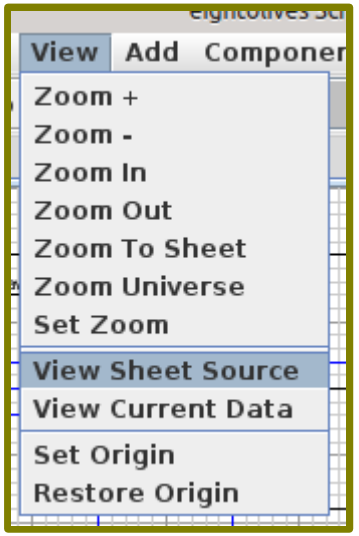
Copyright © 2014 William Kaupinis All Rights Reserved

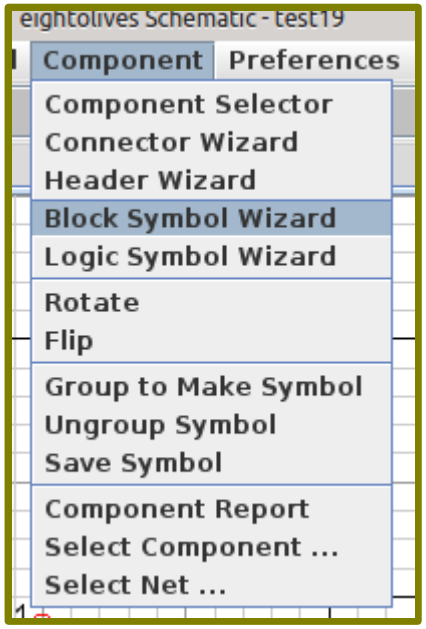
The Menus

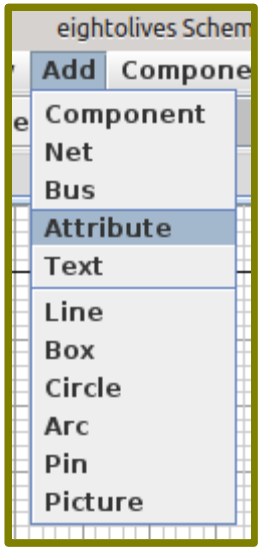


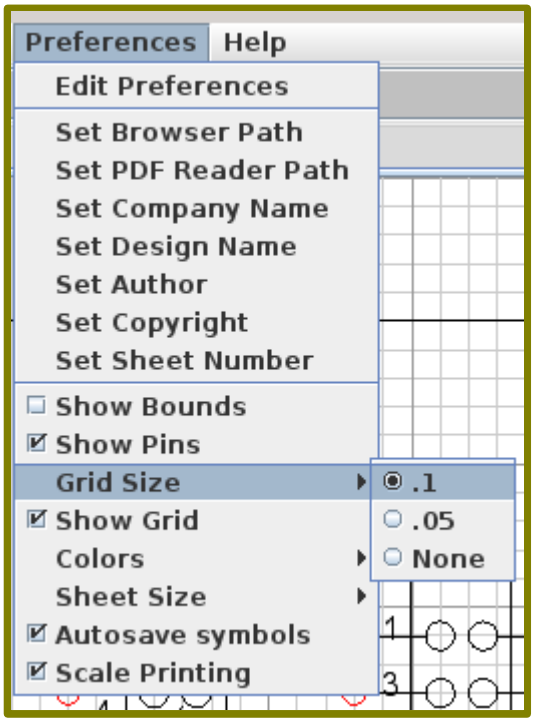


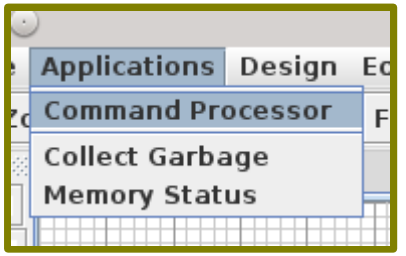


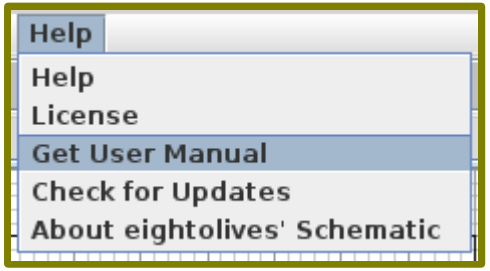












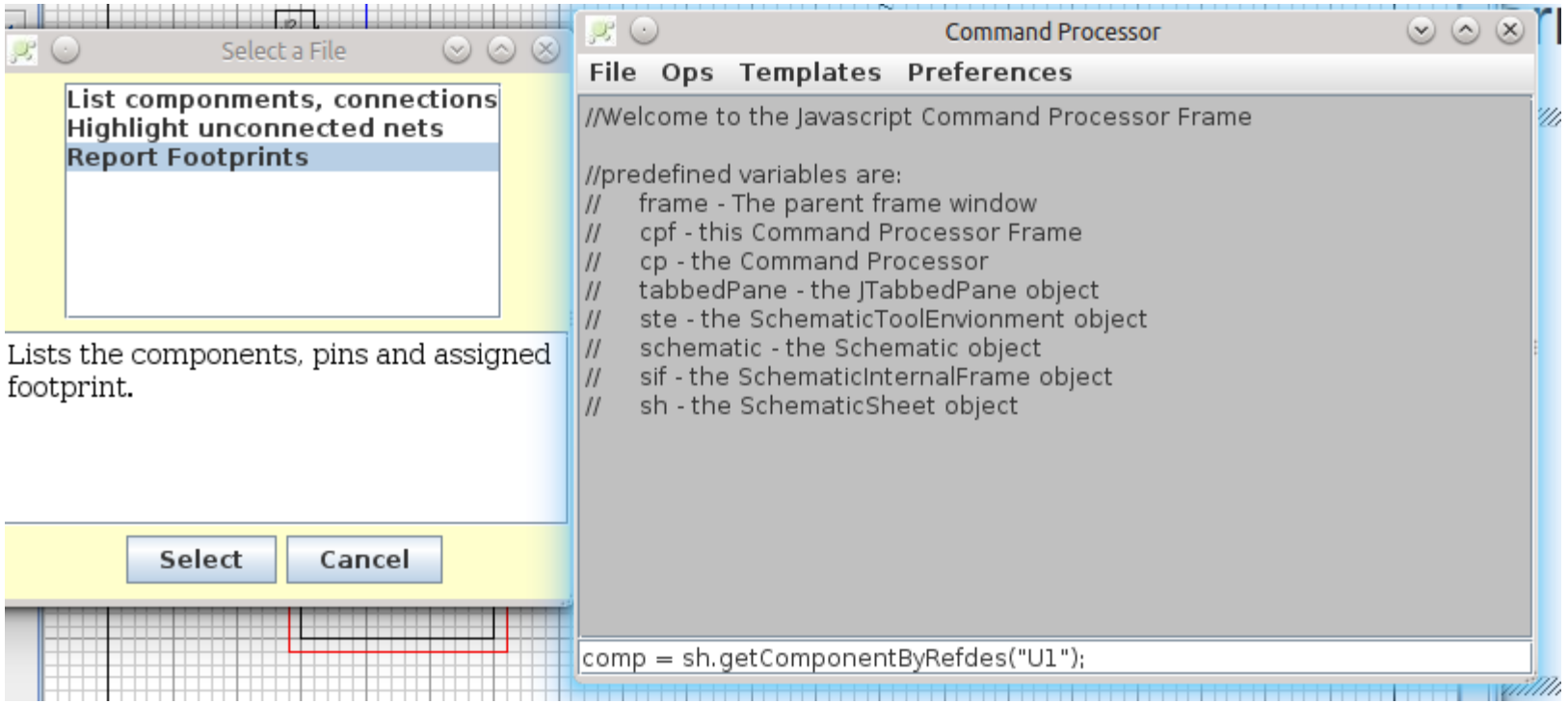
Copyright © 2014 William Kaupinis All Rights Reserved

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, …);

Copyright © 2014 William Kaupinis All Rights Reserved

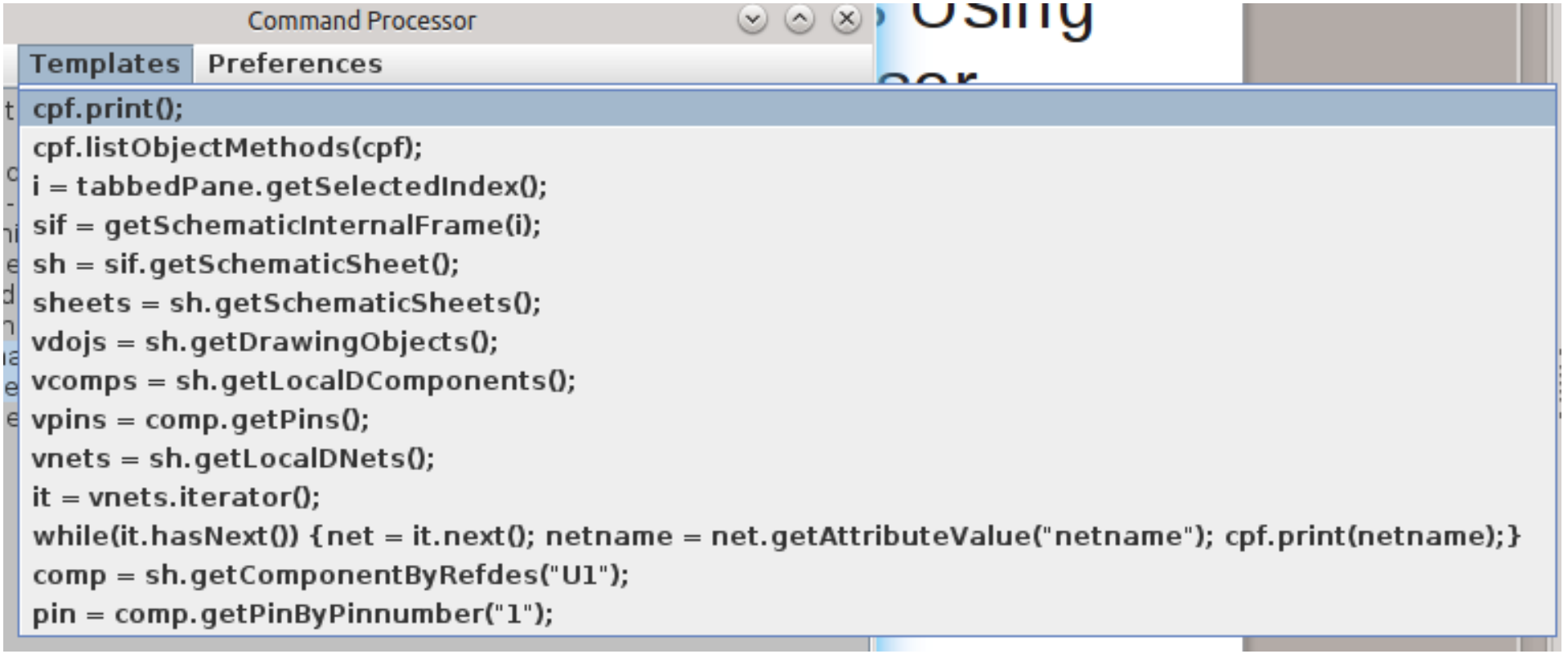
Execute Javascript Scripts Using the Command Processor



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 © 2014 William Kaupinis All Rights Reserved

Templates for often used commands



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

Copyright © 2014 William Kaupinis All Rights Reserved

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

Copyright © 2014 William Kaupinis All Rights Reserved

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

Copyright © 2014 William Kaupinis All Rights Reserved

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

Copyright © 2014 William Kaupinis All Rights Reserved

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

Copyright © 2014 William Kaupinis All Rights Reserved

FAQ

- 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.

Copyright © 2014 William Kaupinis All Rights Reserved

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

Copyright © 2014 William Kaupinis All Rights Reserved