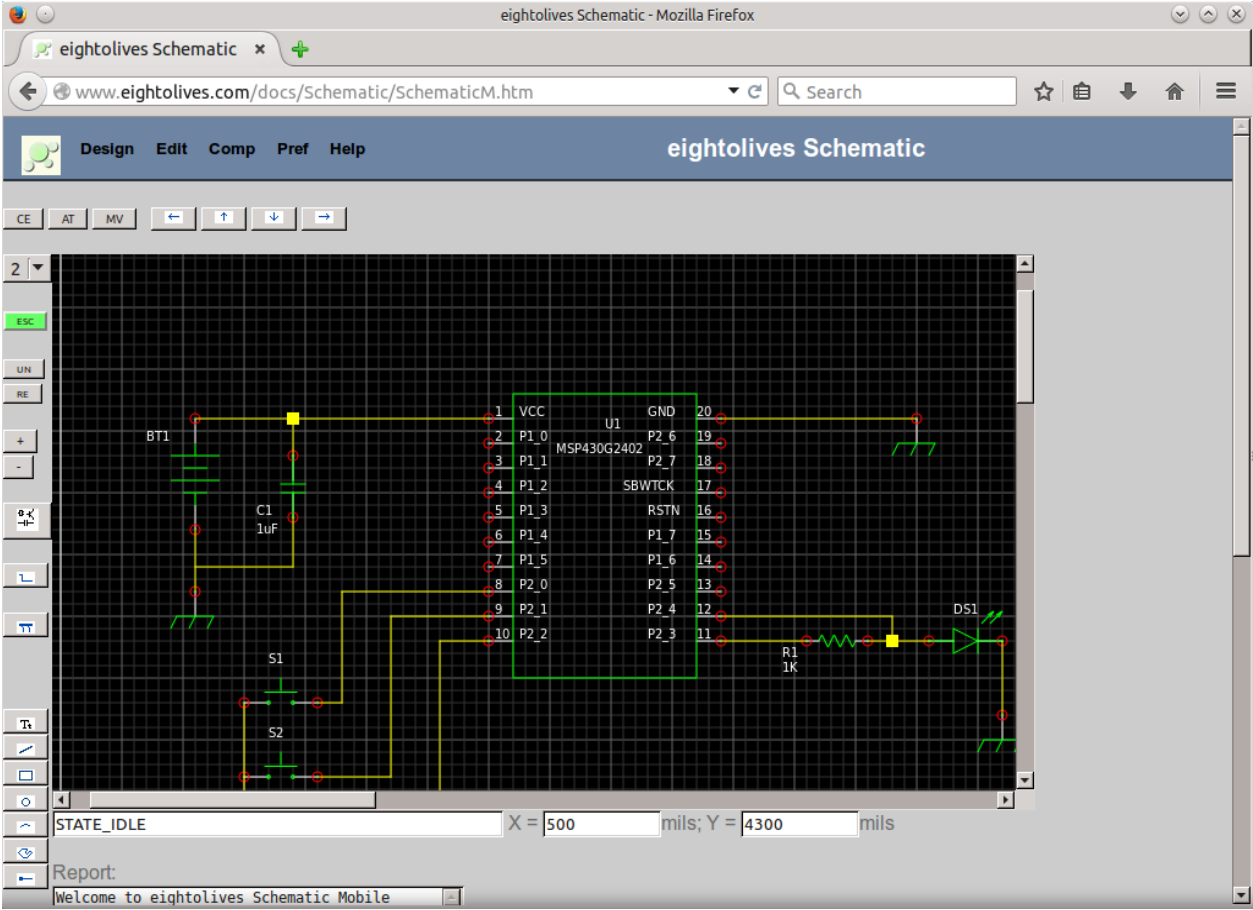




# Using eightolives' Schematic Mobile



- Eightolives' Schematic Mobile is a web-browser-based 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, a database 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 Generate a Netlist or start a simulation or go to printed circuit board layout.

- Menu Bar
- Component Edit
- Attribute Edit
- Move
- Sheet select
- Escape
- Undo
- Redo
- Zoom In
- Zoom Out
- Component Select
- Draw Net
- Draw Bus
- Add Text
- Draw Line
- Draw Box
- Draw Circle
- Draw Arc
- Draw Path
- Add Picture
- Add Pin
- Status Line

The screenshot shows the 'eightolives Schematic' application running in Mozilla Firefox. The browser address bar displays 'www.eightolives.com/docs/Schematic/SchematicM.htm'. The application window has a menu bar with 'Design', 'Edit', 'Comp', 'Pref', and 'Help'. Below the menu bar is a toolbar with buttons for 'CE', 'AT', 'MV', and navigation arrows. The main workspace is a grid with a circuit diagram. The diagram includes a battery (BT1), a 1uF capacitor (C1), a switch (S1), and a 1K resistor (R1). The central component is an MSP430G2402 microcontroller (U1) with its pinout listed in a table:

1	VCC		GND	20
2	P1_0	U1	P2_6	19
3	P1_1	MSP430G2402	P2_7	18
4	P1_2		SBWTCK	17
5	P1_3		RSTN	16
6	P1_4		P1_7	15
7	P1_5		P1_6	14
8	P2_0		P2_5	13
9	P2_1		P2_4	12
10	P2_2		P2_3	11

At the bottom of the interface, there is a status line showing 'STATE\_IDLE' and coordinates 'X = 7000 mils; Y = 8500 mils'. A 'Report:' section at the very bottom contains the text 'Welcome to eightolives Schematic Mobile'.

# Why a schematic?

- A picture is worth 1000 words.
- A schematic is useful to graphically specify the connection of parts leading to a Bill of Materials (BOM) and net list.
- Used in designing printed circuit boards, generating documentation, creating database for circuit simulation.

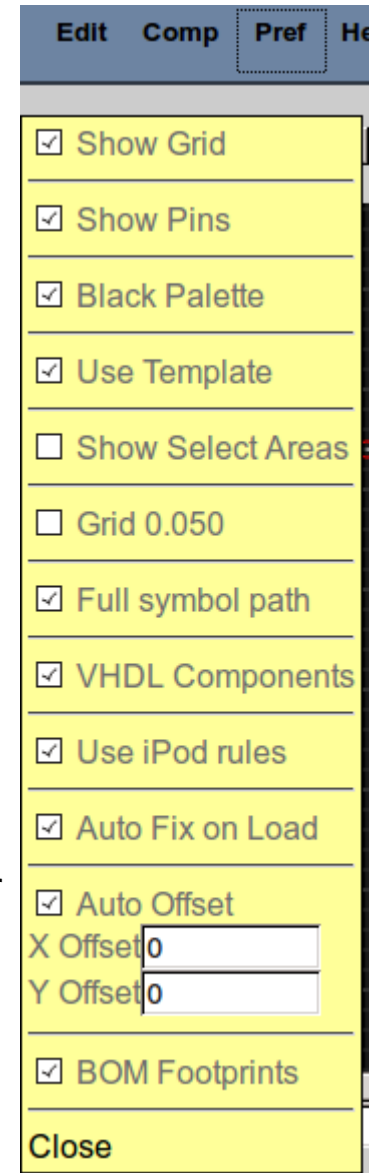
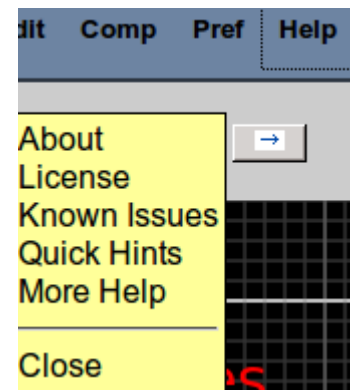
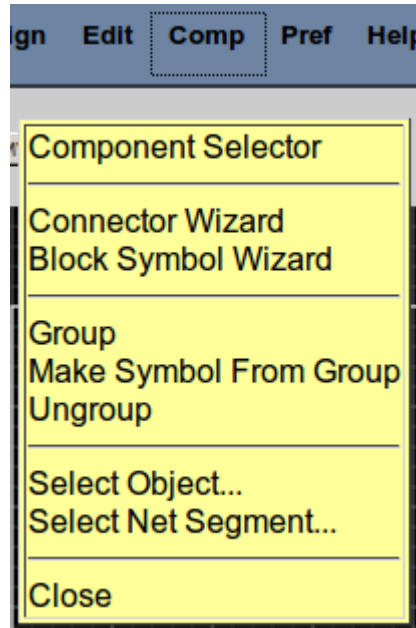
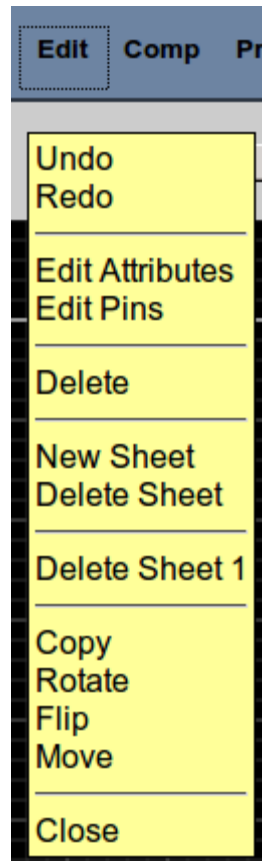
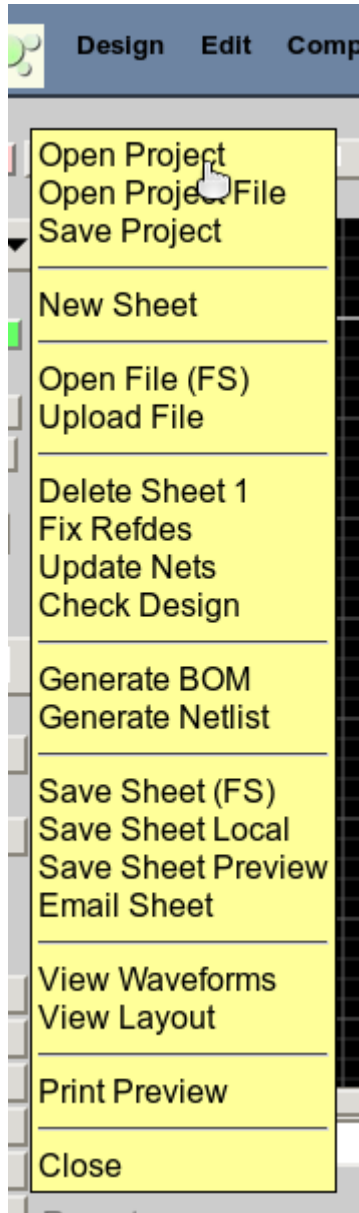
# Basic Definitions

- A **service** is defined as a provider and place of storage, symbols and other features.
- 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.
- An **attribute** is a name/value string pair used to customize information. Many attributes are predefined for various purposes.
- A **net** represents a signal that is connected to pins of the various parts. Net connections can be made of drawn net segments or implied by the use of net stub components or the net attribute.
- A **bus** is an arbitrary group of nets.
- A **library** is an accessible set of symbol files that represent common parts

# Mobile Platform Constraints

- Limited memory and file storage (not so reliable).
  - Use browser “cookie” memory for LOCAL storage
  - Use a companion Microserver to store files on a wifi-networked PC
  - Use Google Drive to access your files from anywhere
- Security concerns of mobile devices and web browsers
  - Companion Microserver uses secure https link, log-in with password
- No mouse
  - Use simple finger actions to easily create schematics
  - **Using a mouse is a lot easier**

# Menu Bar Options



The default, turn-on **project** is called “**LOCAL**” and uses your browser's “cookie” cache memory.

Other projects are accessed via a companion server which supports file storage. The companion server runs on a PC accessible on your local network.

# Services and Projects

Open Project

Select Service:

LOCAL

Server

External service

Drive

Select Project:

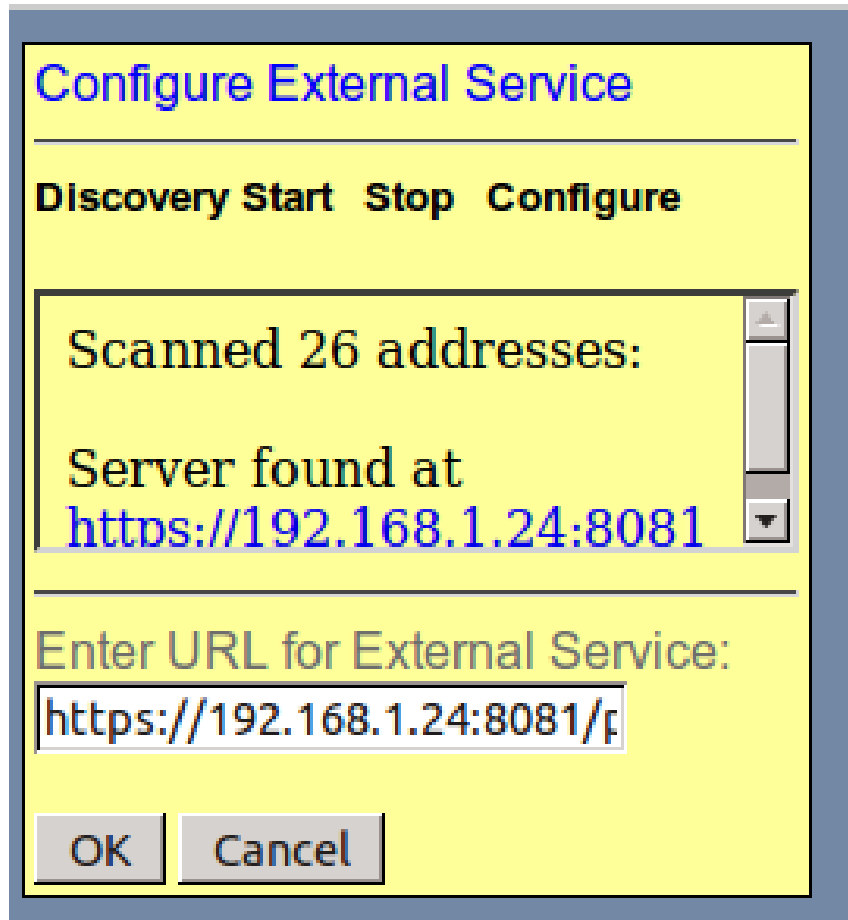
Local Project

Select Cancel

- The default Service is LOCAL which lets you save files in browser cache memory.
- With Server selected, various example Projects may be available from the Schematic Mobile server.
- External Service lets you store files on another server such as a computer running Microserver.
- Drive lets you connect to your Google Drive account.



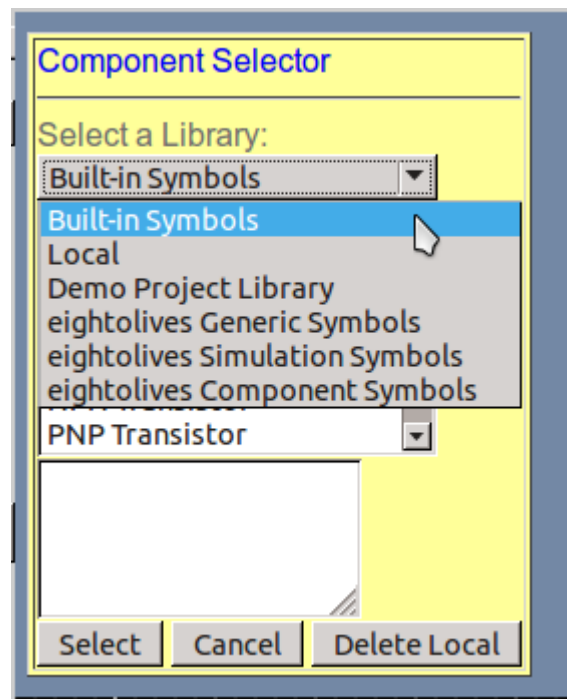
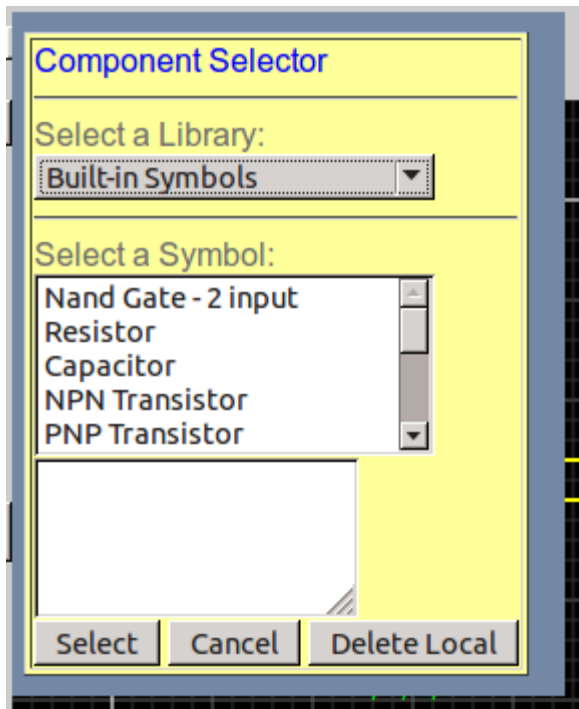
# Configuring External Service



- Currently External Service can be used with an eightolives Microserver operating on your local network.
- You can use the Discovery option to scan for available Microservers
- Clicking the found server enters the URL for External Service

# Pick symbols from Libraries

Use the Component Selector to pick a symbol from a library



A few basic symbols are built-in.

Eightolives provides some symbol libraries that are accessible either from the direct web site version of the tool or via the companion sever version.

Project Library accesses symbols stored fo your selected project.

*Local* accesses symbols in your browser's cache memory.

# Use a Wizard to Make a Symbol

**Connector Wizard**

Reference Designator

Number of Pins

Dual Row Header

Pins on Right

Pins on Left

Header both sides

**Block Symbol Wizard**

Reference Designator

Device Name

Start Pin

Number of pins on Left Side

Number of pins on Right Side

Number of pins on Top

Number of pins on Bottom

Minimum Width (in mils)

Minimum Height (in mils)

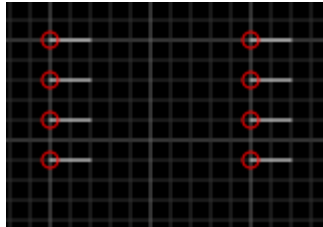
Half Size

You need to save symbols you create. "Save Project" will save both sheets and symbols. The Connector Wizard makes generic connectors and headers. The Block Symbol Wizard makes box-type symbols which you can easily customize.

# Use a Vendor's Symbol

- Part manufacturers may provide symbol and footprint data in the proprietary “.bxl” format
- Use the free Ultra Librarian Binary Reader to read the “.bxl” file and then export it with “XML Library Export” selected
- The resulting .xml file, stored in the “UltraLibrarian\Library\Exported” directory, can be input to Schematic Mobile
  - Note: the exported .xml file is named with a date stamp rather than the component name

# You Can Manually Create Symbols



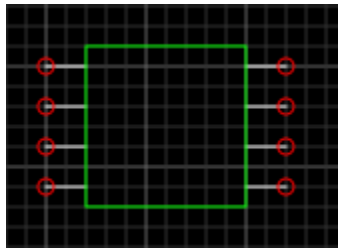
Place Pins on a sheet.

“Group” also works with the move arrows and Delete.

ESC to exit Group.



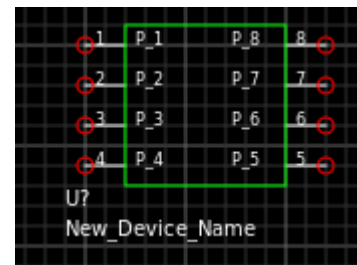
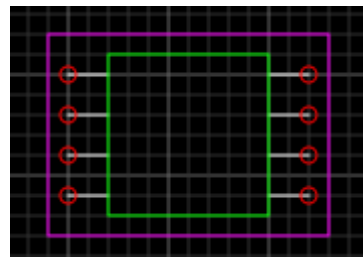
Select each pin on the right side and Flip from the Pin popup.



Draw the graphic rectangle.

Group

then Make Symbol From Group



You can then use the Pin Editor and modify attributes.

Save the symbol!

# Drawing Functions

- The drawing functions can be used in making symbols:
  - Line
  - Box (rectangle)
  - Circle
  - Arc
  - Path (an arbitrary closed shape)
  - Pin (adds a functional pin to a symbol)
  - Picture (an external .png or .jpg picture)

# Select a Part, Net or Pin with a Touch or Left Mouse Click

A second touch or click pops up a menu of options

- Edit Attributes
- Edit Component
- Edit Pins
- Toggle Pin Numbers
- Add Net to Pin...
- Add Net Stub to Pin...
- Add Net Stubs All
- Select Pin...
- Check / Fix Dupe Names
- Save Symbol (FS)
- Save Symbol Local
- Save Symbol Preview
- Email Symbol
- Save Symbol Project
- Report Sim Template
- Make Comp Schematic
- Copy
- Move
- Move Ref
- Move Value
- Rotate
- Flip
- View Documentation
- Set Footprint
- Set Placeneer
- Delete
- Close

- Edit Attributes
- Edit Net Segment Name
- Net Comment
- Adjust Net
- Add to Waveforms
- Move
- Delete
- Close

- Edit Attributes
- Add Net Stub
- Add Net
- Close

- Rotate
- Flip
- Move
- Delete
- Close

**Pin Selector**

Select a Pin:

1: CLK

Select Cancel

**Footprint Selector**

Select a Library:

gEDA Footprints

Select a Footprint:

TO251, diode in TO220 TO2...  
 TO264, diode in TO220 TO2...  
 TO39, Transistor TO39.fp  
 TO92, Transistor TO92.fp  
 TQFN12\_4\_EP, Square Qua...

Details:

T092,  
 Transistor  
 T092. fp

Use Selected

Footprint:

TO92.fp

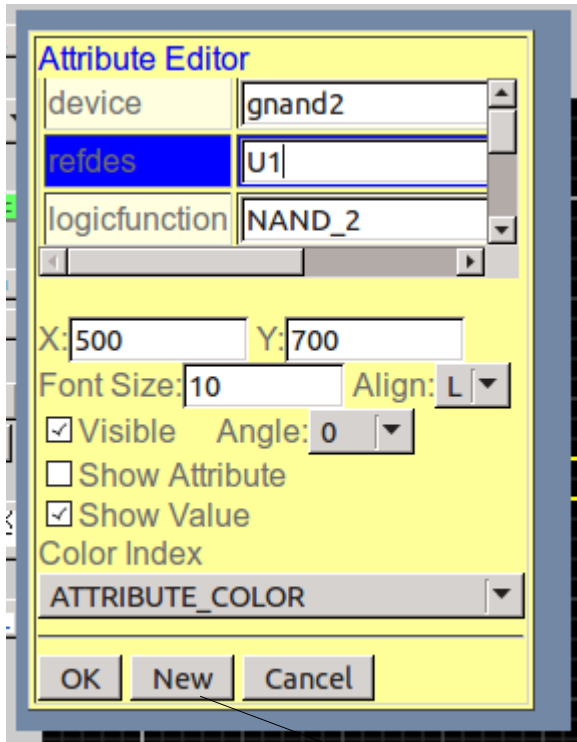
Set Cancel

Touching or clicking in empty schematic area closes the menu and unselects the item.

Edit Attributes is also invoked from the top "AT" button.

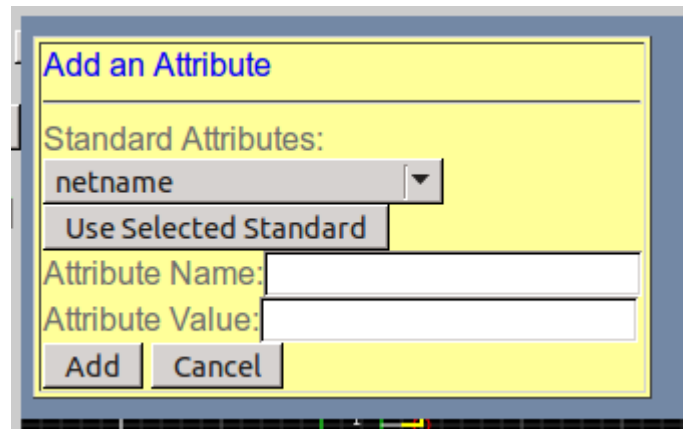
Edit Component is also invoked from the top "CE" button.

# Attributes Personalize Symbols, Pins and Nets



Symbols, pins and nets are created with certain, mandatory attributes. A net has an attribute called “netname”. A component has attributes called “device” and “refdes”. A pin has attributes called “pinlabel”, “pinnumber”, “pinseq” and “pintype”.

You can add other standard or non-standard attributes. Some attributes have special meaning or affect other tools you may use. See the manual for details.





# A few words on 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
- placenear – layout hint to place near refdes-pin (e.g. U2-7)
- net – defines hidden pins sometimes used for power and ground connections

- **For nets:**

- netname – the name of the net

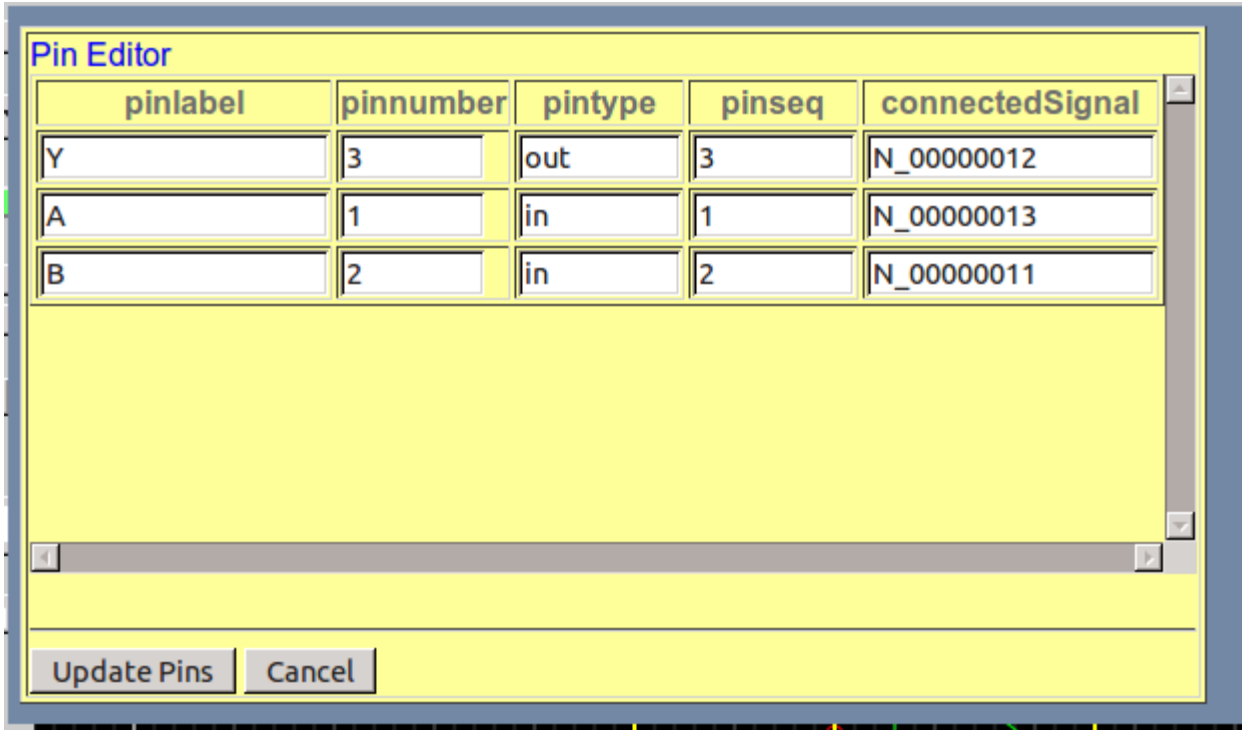
- **For pins:**

- 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

# Special eightolives 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 characters is interpreted as timing parameter such as Tdelay or Tperiod. Values should be a constant time value such as 10 ns
- **src** – a full URL to access a symbol

# Edit Pin Information Easily



The Pin Editor dialog box is shown with a yellow background. It contains a table with the following data:

pinlabel	pinnumber	pintype	pinseq	connectedSignal
Y	3	out	3	N_00000012
A	1	in	1	N_00000013
B	2	in	2	N_00000011

At the bottom of the dialog box, there are two buttons: "Update Pins" and "Cancel".

The Pin Editor lets you edit a component's pin information in a table form.

The connectedSignal is the current net connection for the pin.

Nets are automatically given unique names with the format Nxxxxxxx which you can rename.

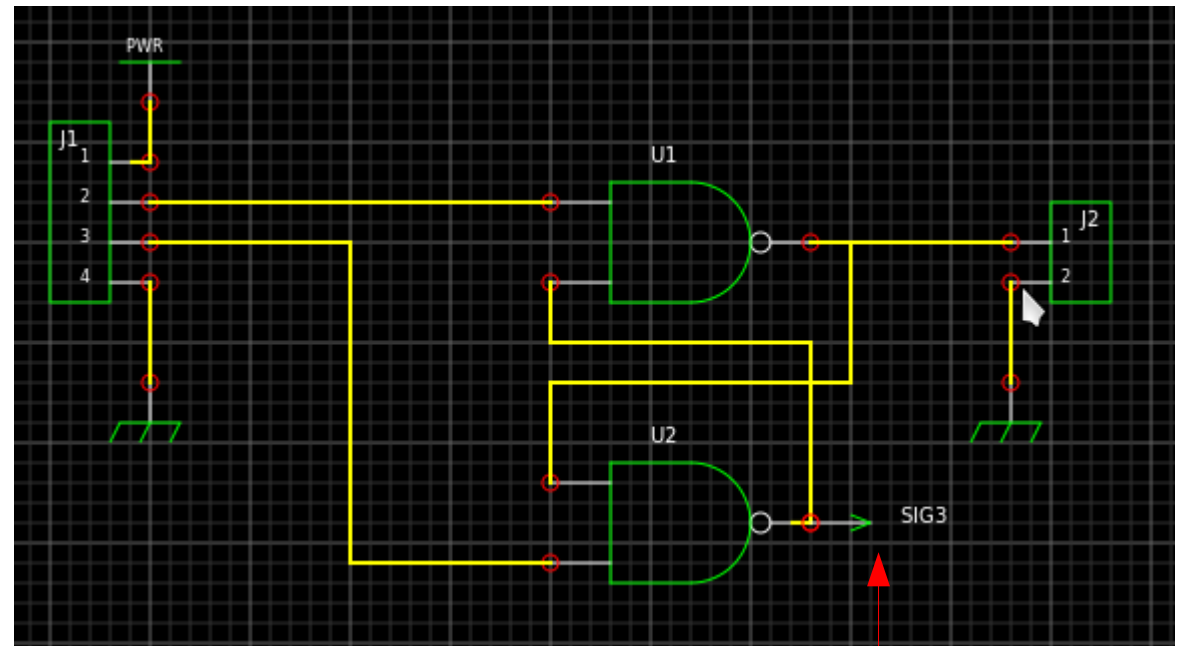
# Nets can be created in different ways

You can select a pin and start drawing a net to another pin.

You can select a component and select the “Add Net to Pin” option to start.

You can select a component and select the “Add Net Stub to Pin” option to use a net stub or off-sheet reference component to connect to a net.

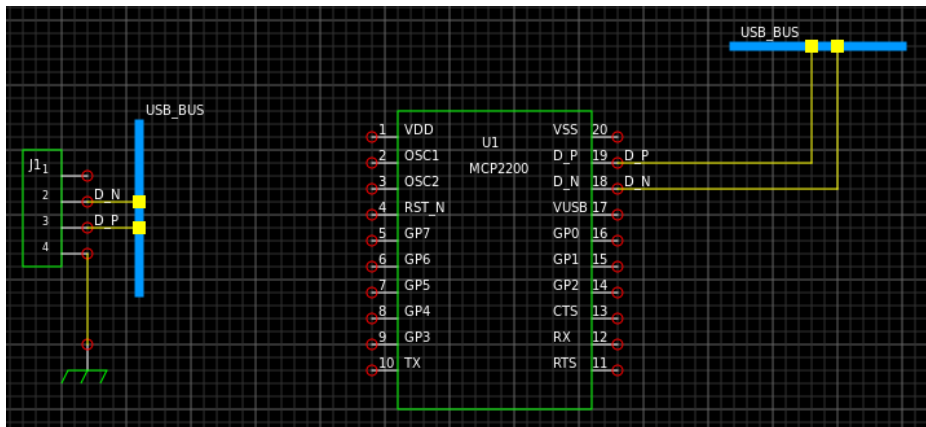
You can use a power or ground symbol whose single pin connection has the “net” attribute which connects to the net.



A Net Stub

Note:  
“Rubberbanding” is not supported

# A Bus is just a set of signals



- A bus is an optional drawing convenience
- Nets connected to pins are the important items in making a netlist
- Nets connected to buses make the net a bus member

- Symbols can be made with “Bus Pins” but it's not gEDA compliant
- See separate paper on Using Buses

# Tools and Reports

## The Typical Operating Sequence:

- Fix refdes
- Update Nets
- Check Design
- Generate Netlist – gEDA, PADS, KiCad, VHDL format or eightolives Hardware Model
- Generate BOM

```
BB Q1-B R1-2
GND Q1-E J2-3 J1-3
N_00000000 U2-3R1-1
N_00000001 U1-3U2-1
N_00000002 U1-1U1-2J2-1
N_00000004 U2-2J2-2
N_00000007 Q1-C R2-2J1-2
PWR R2-1J1-1
```

Check Design 12/17/14

Net Checks:  
Nets OK

Check Sheet 12/17/14

Design\_Name  
1: SHEET 1 OF 2

Check Sheet 1 OK  
No warnings

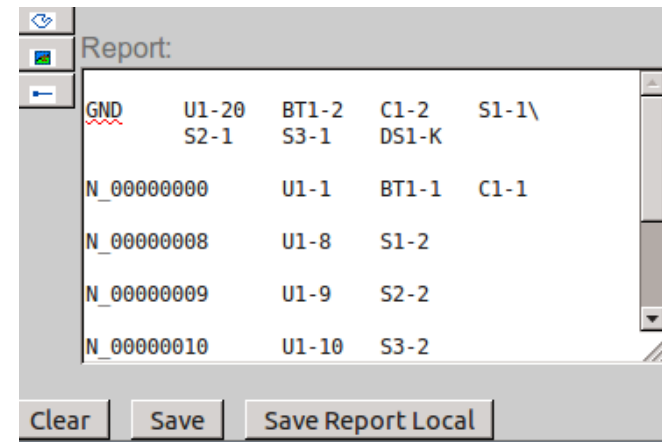
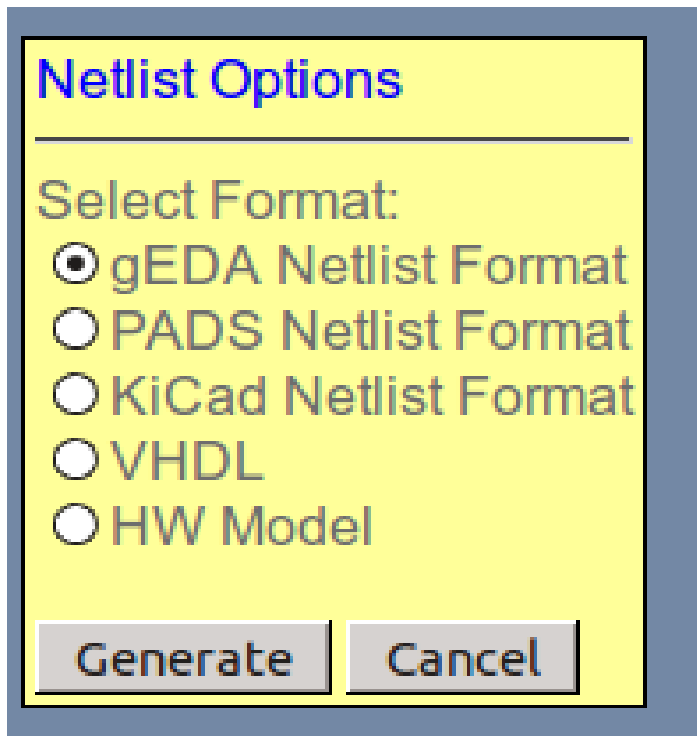
Check Design OK  
No warnings

// Bill of Materials Design\_Name 12/17/14

// eightolives  
// DRAWN BY:  
// copyright (c) 2014 eightolives

QTY	PART ID (DEVICE)	REFERENCES
1	CONNECTOR_3L	J1
1	CONNECTOR_3R	J2
2	gand2	U1, U2
1	NPN	Q1
2	RES	R1, R2

# Generate Netlist



- Netlists are written to the Report window. You can then save or copy the netlist from there.

# Using the File System (FS)

- Browsers running on Windows or Linux permit manual access to the platform's File System.
  - You can manually store and retrieve schematic sheets and symbols to the directory of your choice
- The iPod does not support a similar File System
- When reading a schematic sheet that requires a symbol that cannot be found or one that is stored in the FS, you will be prompted to locate the symbol which is then loaded to LOCAL project
  - Reloading the schematic will use symbols from LOCAL



# gEDA Compatibility

- For open issues use menu option Help > Help and select *Schematic Mobile Open Issues*
  - <http://www.eightolives.com/docs/Schematic/SchematicMobileIssues.htm>
- gEDA drawings are normally centered in a larger drawing universe
  - *Pref > Auto offset* automatically moves the gEDA A size drawing to Schematic Mobile's A size universe (gEDA's lower left corner)
- Schematic Mobile defaults to storing a full path to symbols (*Pref > Full Symbol Path*)
- gEDA uses stored symbols from a shared directory structure. Schematic Mobile saves symbols to the project directory

# gEDA Compatibility

- Each schematic sheet is saved as a separate file.
  - Schematic Mobile can also input a single file which is a concatenation of all sheet files
    - `cat design1.sch design_2.sch design_3.sch > design.sch`

# Support for Simulation

- Digital simulation with eightolives' WaveformViewer is supported with several menu command options
  - Design > Generate Netlist > HW Model
  - Double click a component > Report Sim Template
  - **Design > View Waveforms**
  - Double click a net > Add to Waveforms
  - Select a net to see its simulation logic value
- See separate papers on using WaveformViewer
- Output a VHDL netlist for other simulators

# Support for Layout

- Support is provided to transfer the design to eightolives' Layout tool.
- For components, you can specify the *footprint* and *placenear* attributes
- From a selected Net Menu, you can select *priority\_route* for the net
- Top level attributes *gnd\_plane\_signal* and *power\_plane\_signal* can specify the signal on those special layers
- Layout can be invoked from *Design Menu > View Layout*
  - The Hardware Model is created and transferred to Layout

# Getting Started

- Schematic Mobile is web page which is a Javascript program that runs in your Firefox, Safari or Chrome browser. It uses “cookie” memory to store your LOCAL project
- You can run the program from a link on the eightolives Home Page or the eightolives QuickApps menu
  - Access to eightolives on-line symbol libraries
  - <http://www.eightolives.com/docs/Schematic/SchematicM.htm>
- Improved file services require access to an external server
  - Google Drive provides an excellent way to access your files wherever you go
  - A companion eightolives Microserver running on a networked computer can provide access to other libraries and store your files on that computer
    - See separate paper on using Microserver with Schematic Mobile
  - Other servers can be used but they must have software that implements the Schematic Mobile server API.

# FAQ

- What format is used for schematic and symbol files? gEDA
- 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 and accessible. Check the bug database for any compatibility issues.
- Can I generate other netlist formats? You can create netlists in gEDA, PADS, KiCad or VHDL format. For other formats, you can use the gEDA tool gsch2pcb tool which supports many formats.
- Can I simulate digital designs? See separate papers on simulating with WaveformViewer.

# For more information

- Visit <http://www.eightolives.com>
- For the gEDA Project Wiki visit <http://wiki.geda-project.org>