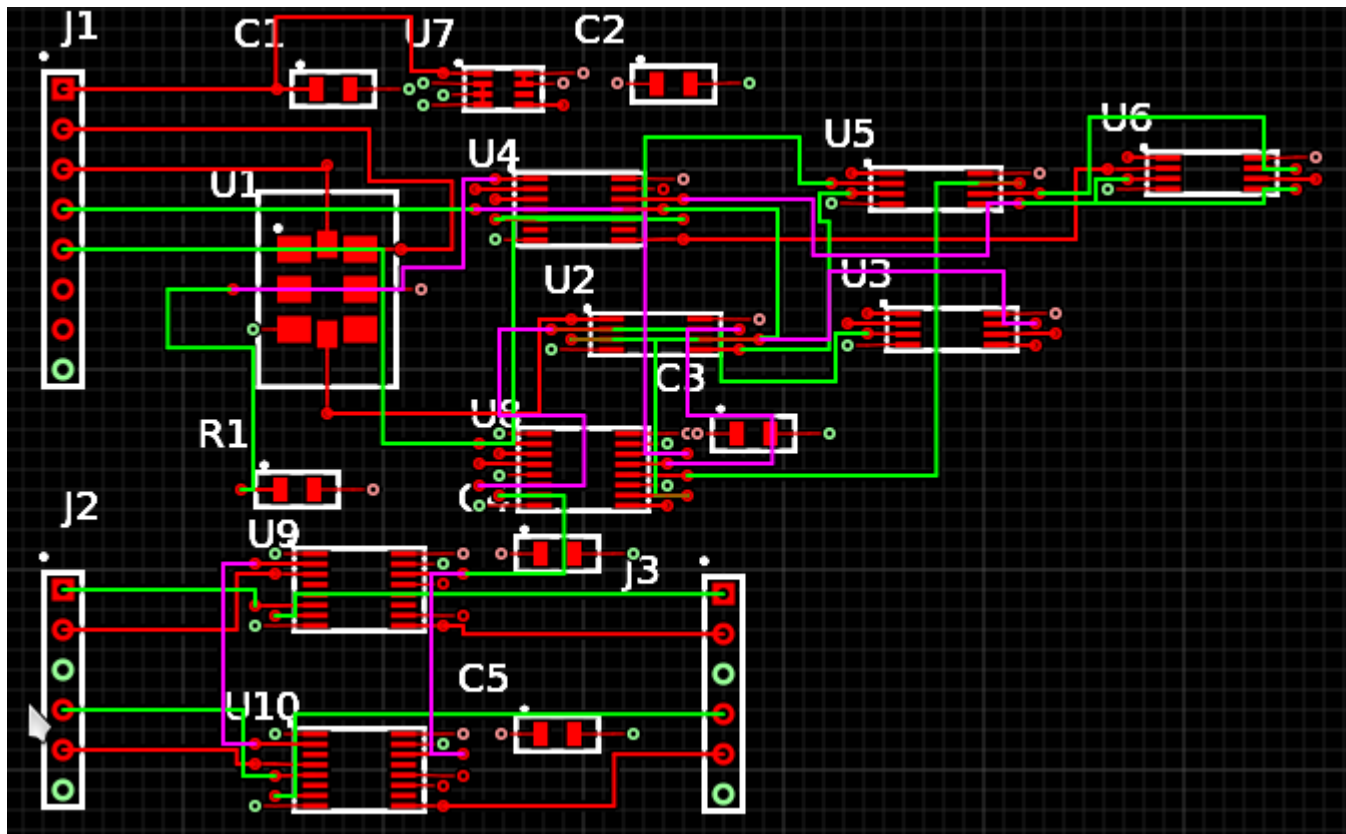




Using eightolives Layout



A web tool for circuit board layout.

William_Kaupinis@eightolives.com
Sept 20, 2016

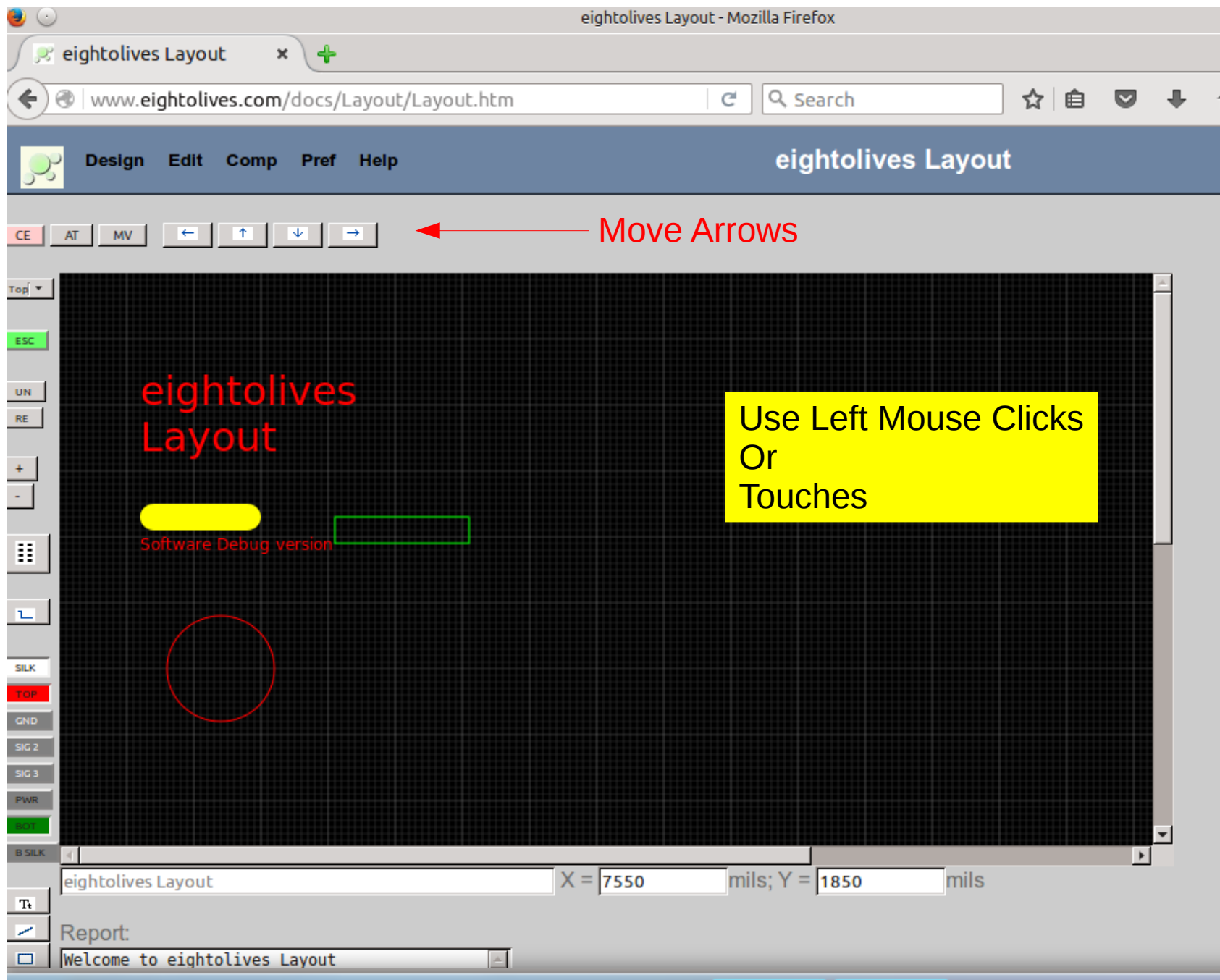
Copyright © 2016 William Kaupinis All Rights Reserved

eightolives Layout Tool

- eightolives Layout is browser-based engineering tool for creating printed circuit board artwork files
- You can create a design directly by selecting footprints from a library and adding traces
- Or you can import a hardware model design created in Schematic Mobile or import gEDA .pcb files
- Save and input the database as gEDA .pcb files
- Output RS-274-X (“Gerber”) files and or .png image
 - Excellon drill file option (deprecated)

Layout

- Create boards with 1-6 copper layers
- Create boards up to 8 by 10 inches
- You can specify design and manufacturing rules
- Layout is a web-based, Javascript tool that runs in your web browser (Firefox, Safari, Chrome)
 - <http://www.eightolives.com/docs/Layout/Layout.htm>
- Store data in Local Project, an External server (such as eightolives Microserver), or Google Drive
 - Can only manually interact with local File System



Top Menu Items
Logo scroll report
CE (pink key)
AT – attributes
MV – move
Active Layer
Escape (green)
Undo
Redo
Zoom + -

Footprint select
Draw a Net

Visible Layer
Toggles

Status Window
Drawing Buttons
Report Window

← Move Arrows

Use Left Mouse Clicks
Or
Touches

Scrolling Down....

Visible Layer

Toggles

Silk

Top

GND

Sig 2

Sig 3

PWR

Bottom

Bottom Silk

Drawing

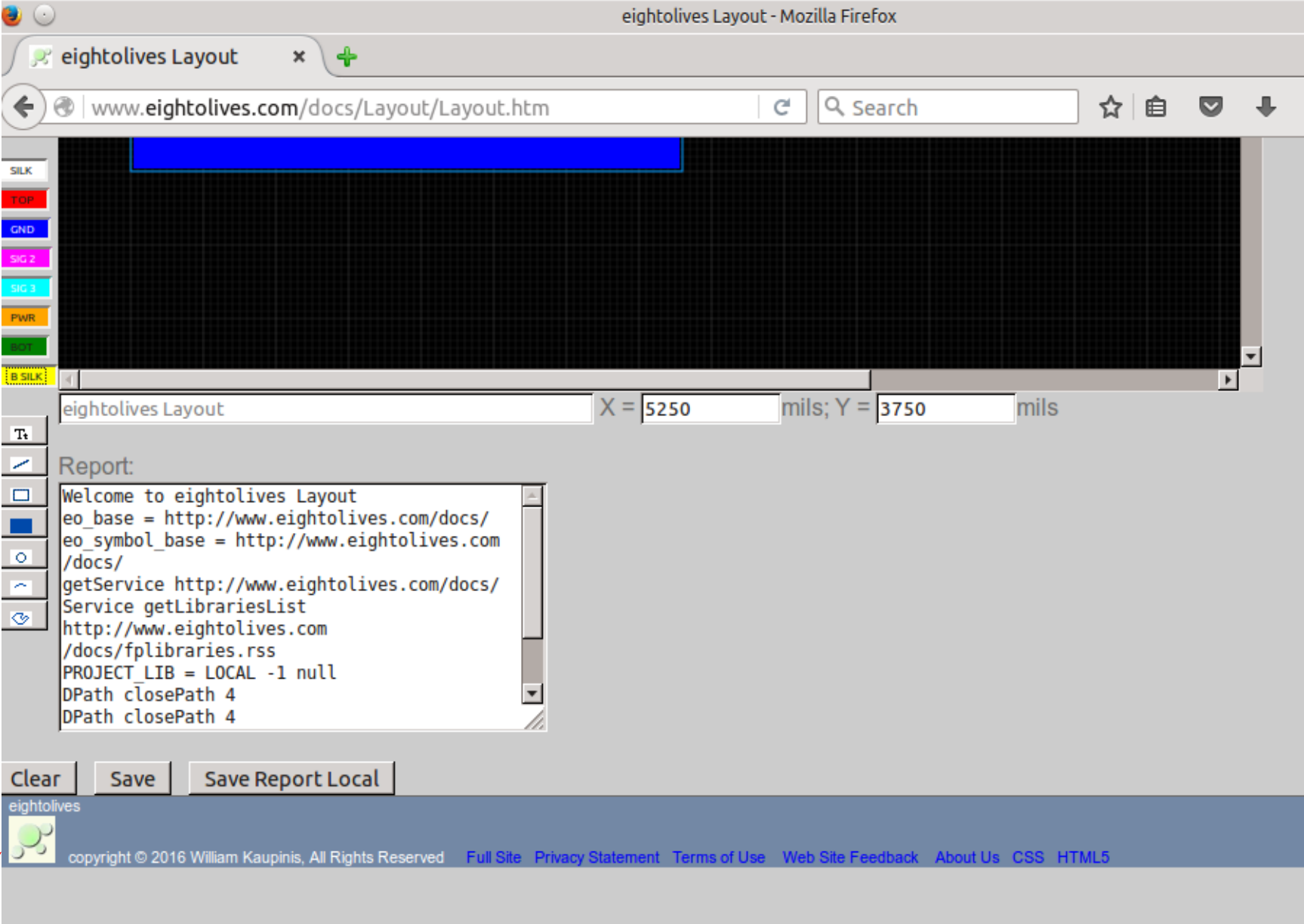
Function

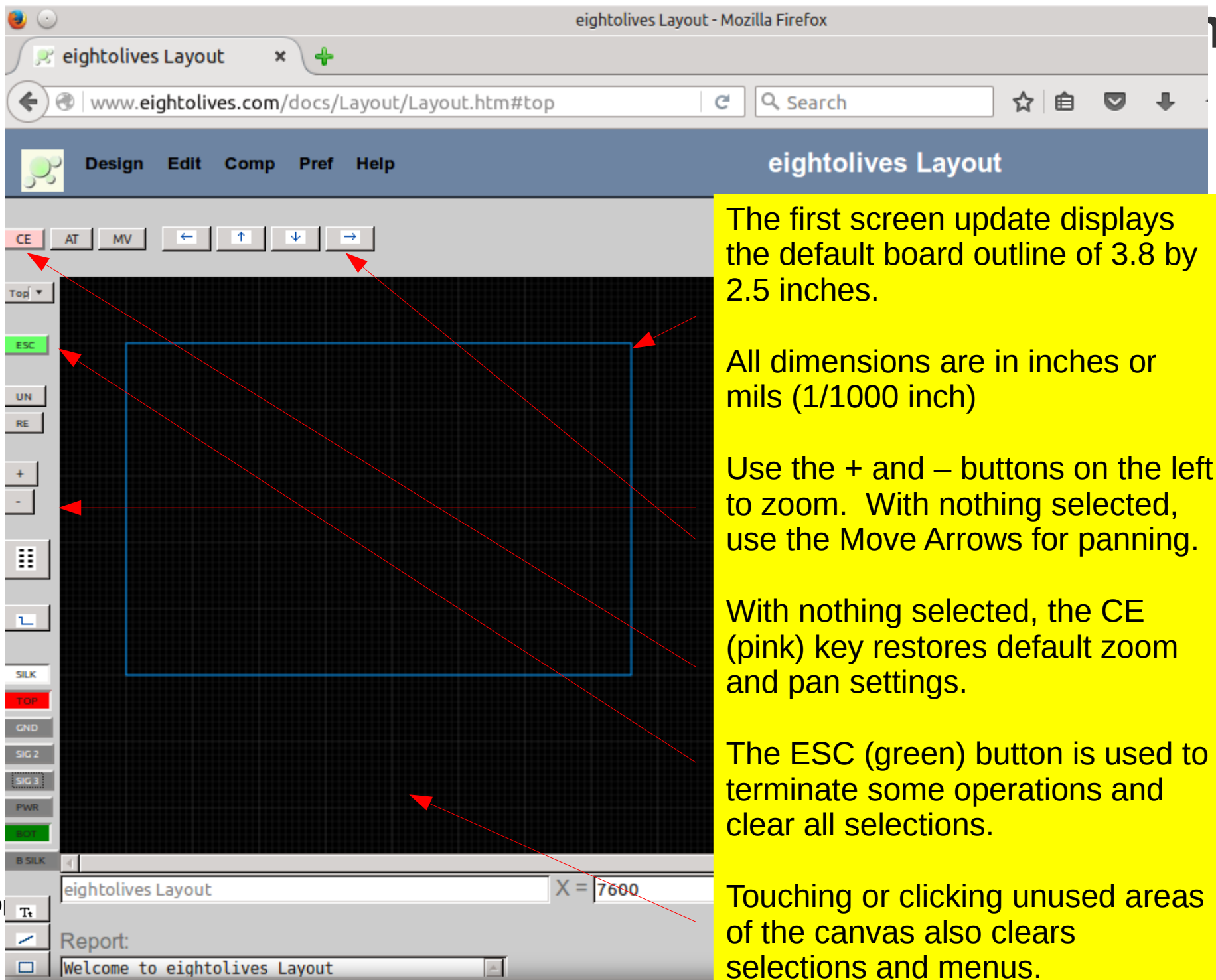
Buttons

Report

Window

Logo scrolls
To Top





The first screen update displays the default board outline of 3.8 by 2.5 inches.

All dimensions are in inches or mils (1/1000 inch)

Use the + and - buttons on the left to zoom. With nothing selected, use the Move Arrows for panning.

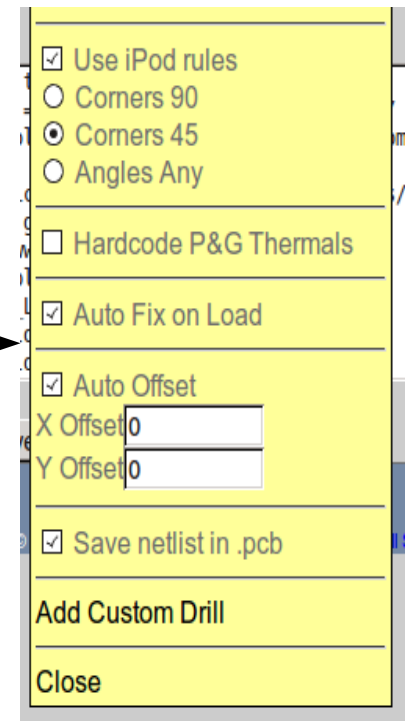
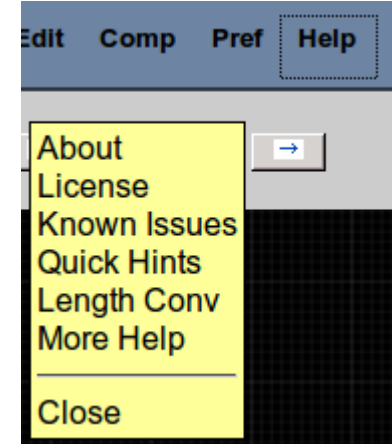
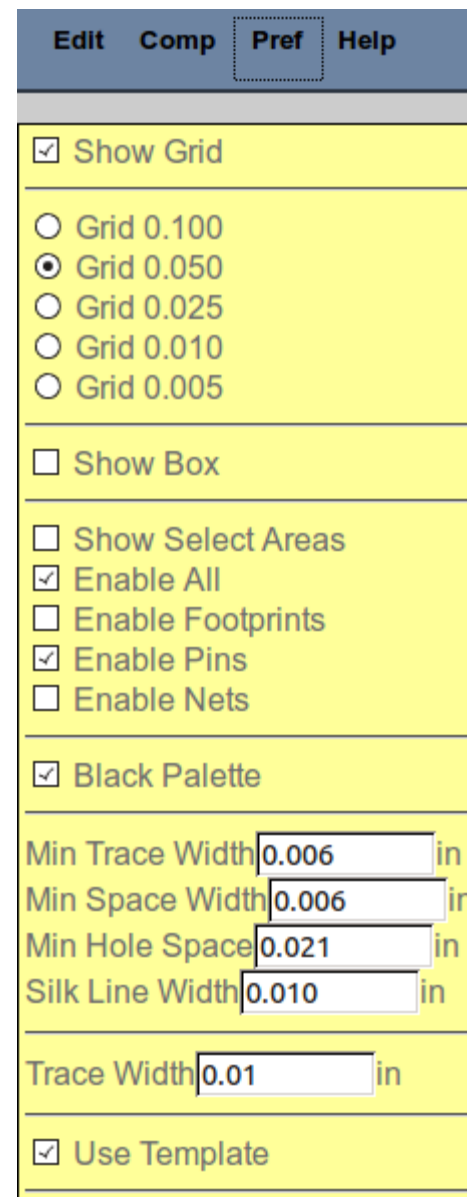
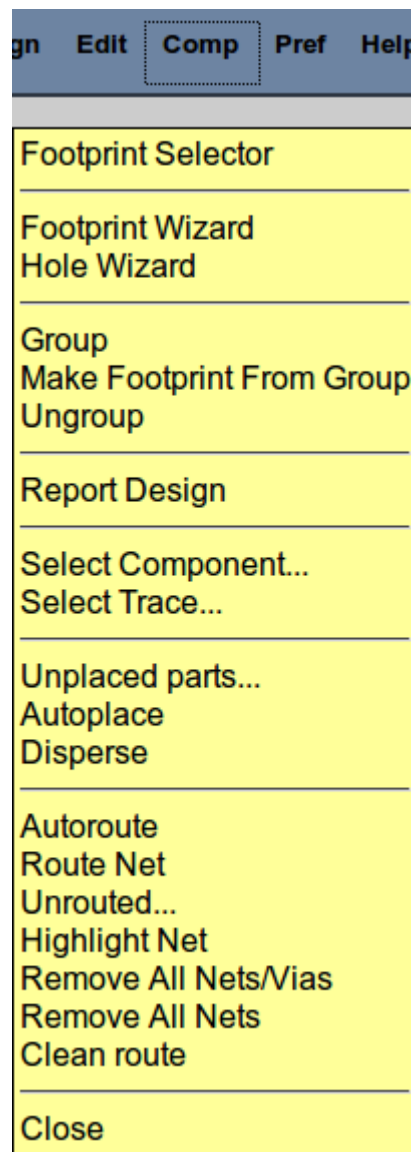
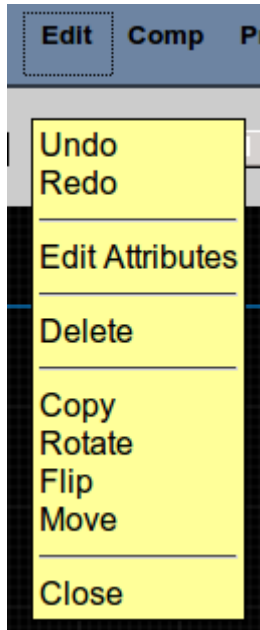
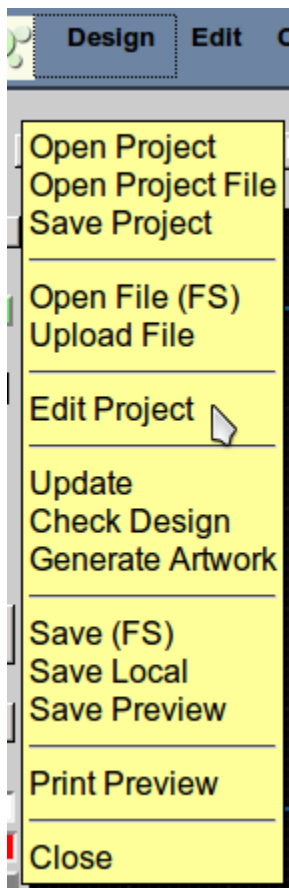
With nothing selected, the CE (pink) key restores default zoom and pan settings.

The ESC (green) button is used to terminate some operations and clear all selections.

Touching or clicking unused areas of the canvas also clears selections and menus.

Top Menu Items

eightolives.com



Design > Edit Project

Project Editor

Project Name

Board width in.

Board height in.

Special shape

Layers

Top silkscreen

Top Layer (signal)

Bottom Layer (signal)

Ground Layer (ground plane)

Ground Plane Signal

Power Layer (power plane)

Power Plane Signal

Bottom silkscreen

Default trace width mils

Default wide trace width mils

Default silkscreen width mils

Minimum clearance mils

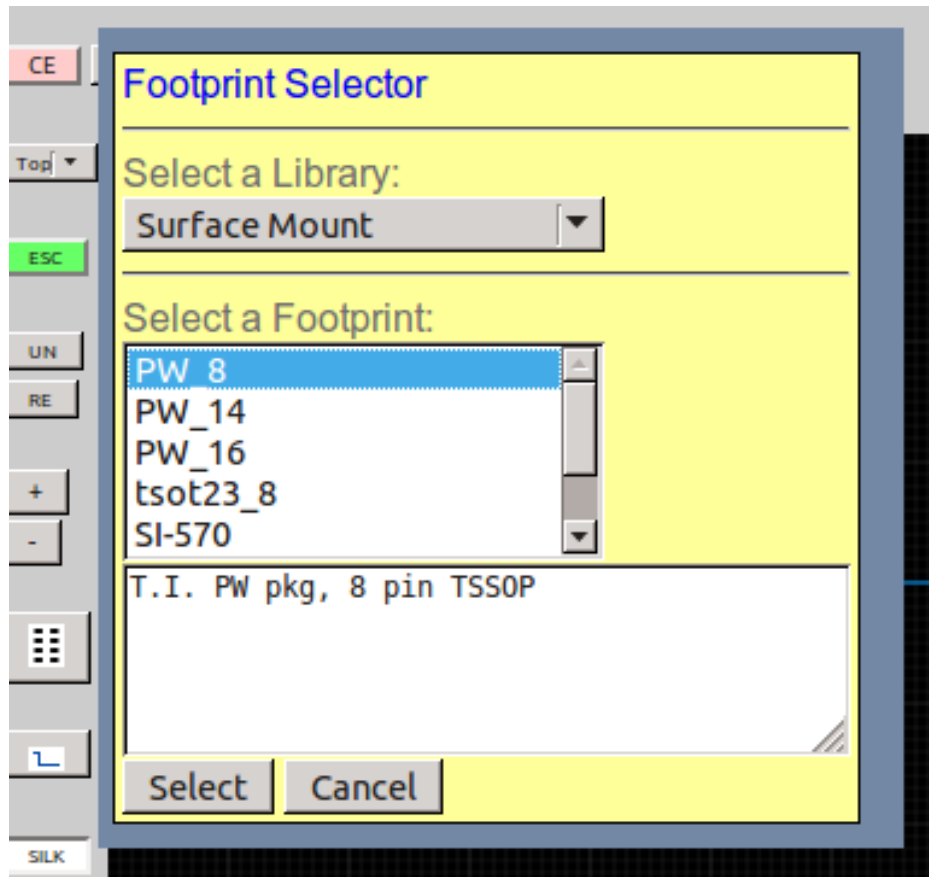
Default via mils

- Use the menu Design > Edit Project to specify
 - Design file name
 - Board dimensions
 - Power and Ground plane signal names

Footprints

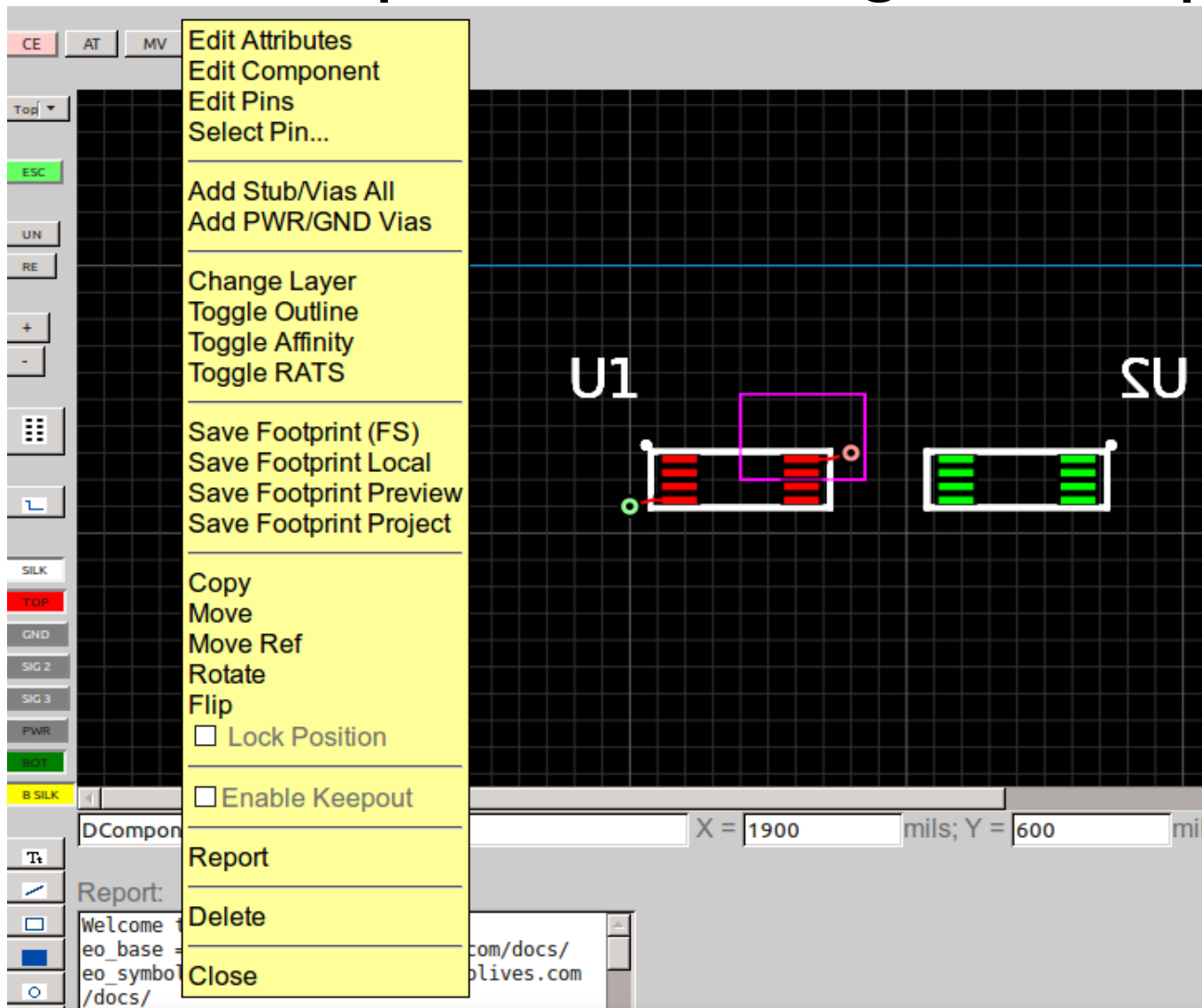
- A footprint describes the pads, holes and vias associated with mounting a component
- Use footprints (.fp files) from the eightolives on-line library or from gEDA libraries
- Create your own pads and footprints by using the Footprint Wizard or just by using the drawing tools.
- Use vendor-supplied “.bxl” files to create .fp footprint files using Schematic Mobile

Footprint Selector



- Select footprints from
 - The Built-In Library – basic resistors, caps, transistors, vias, holes
 - Eightolives' on-line Surface Mount and Through-Hole and Connector libraries
 - Local Project

Select a part twice to get Component Menu



U1 on top layer (red) with stubs and vias connected to power and ground planes.

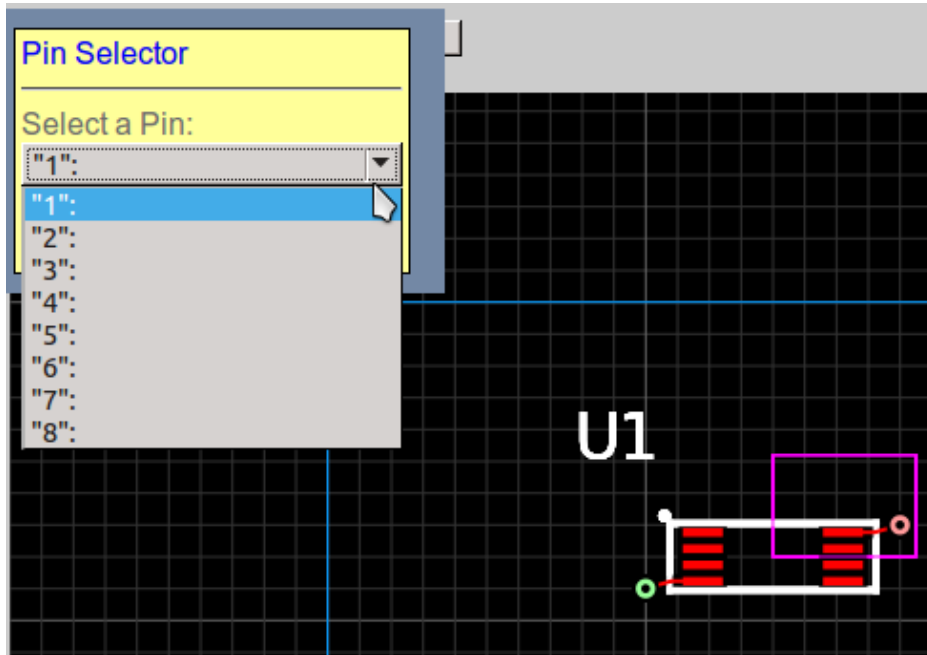
Violet box is the “select area”

U2 is on bottom layer (green) as a result of the Change Layer menu option.

To select a pin, click the Select Pin... option. Then select the pin number from a drop-down menu.

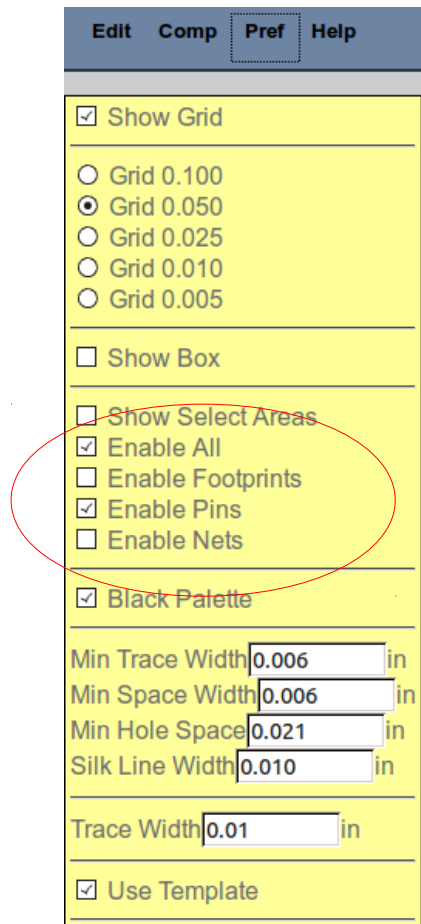
Preferences menu has other options to “filter” what you want to select.

Selecting a Pin



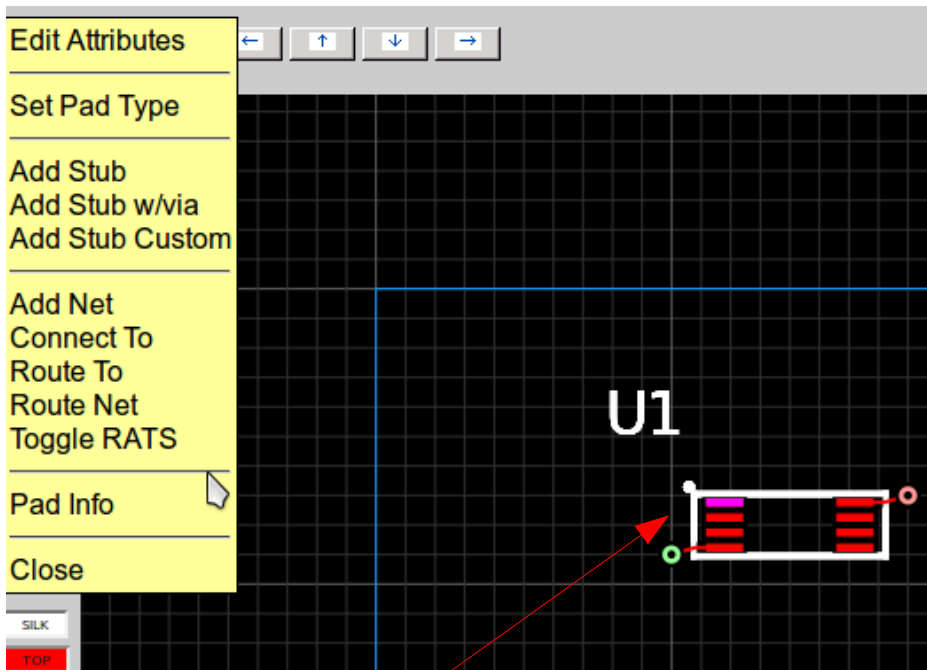
- The drop-down menu lists pin numbers with other info if available.
- You can also select a pin directly if accurately clicked or zoomed-in

Pref Menu Has Select Filter Options



- Enable All – allows footprints, pins and nets to be selected
- Enable Pins – allows only pins to be selected

Pin Menu



Selected Pin is highlighted.

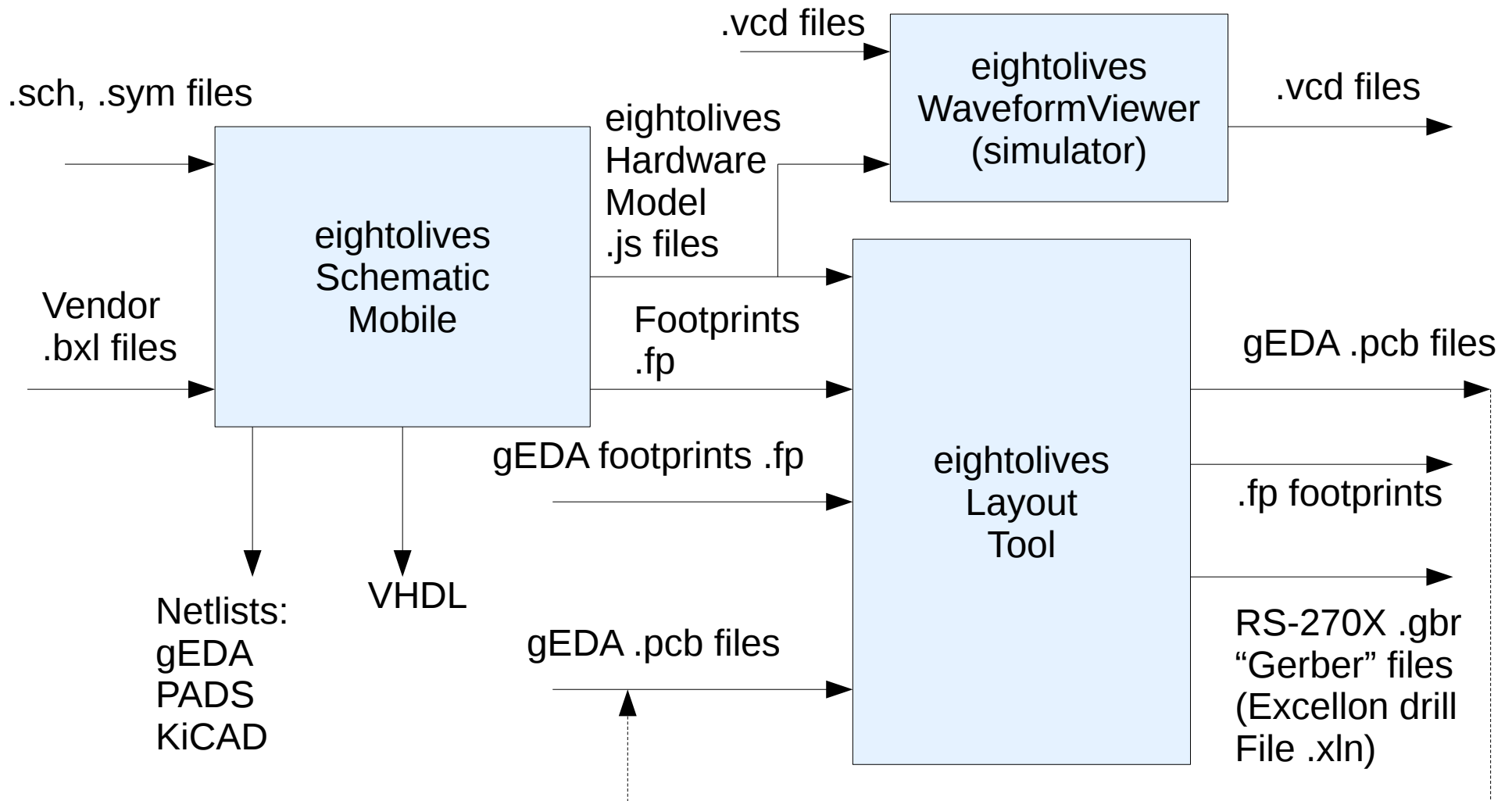
- Adding a stub or stub with via to a surface mount pin adds a wire from the pin center to a “safe” point on the active grid to allow vias to be safely placed.
- Set Pad Type lets you define “thermal” connections on vias and through-hole pins.
- Add Net lets you manually add a net.
- Connect To lets you specify destination (but still manual)
- Route To lets you specify destination (automatic)
- Route Net automatically connects all pins on the net

Design Flow

Schematic Mobile Design
Get / make footprints
Load design in Layout
Place / Autoplace
Route / Autoroute
Check Design
Gen Artwork Files
Check Artwork Files

- The recommended Design Flow starts with creating a design in Schematic Mobile
- Make sure footprints exist for all parts
- Load the design in the Layout tool
- Place parts in your board outline
- Route the design
- Check the design
- Generate the Artwork files
- Check the Artwork files

File Interfaces



Design Using Schematic Mobile

- Schematic Mobile creates Hardware Models (.js files) used by Layout. Hardware Models contain schematic, component and net interconnect data.
- In Schematic Mobile:
 - Assign components' footprints using the **footprint** attribute
 - Assign the **placenear** attribute to certain parts as guidance for Autoplace (eg placenear U2-16)
 - Set the attributes **gnd_plane_signal** and **power_plane_signal** to the appropriate signal names
 - On critical nets, **Set Net Priority Route** from the net popup menu
- In Schematic Mobile, to start Layout use menu option *Design > View Layout*

Netlist Data

- Netlist data is loaded via .js Hardware Models created by Schematic Mobile
- Saved .pcb files can also contain netlist data which is used if no .js file was previously loaded

Footprint Wizard

Footprint Wizard

Footprint Name

Pad Type

Pad Width (V height) in

Pad Length (H width) / diameter in

Hole radius

Number of pads:

Left Side / Rows

Right Side

Top / Columns

Bottom

Pad pitch center-center in

Left-right center-center in

Top-bottom center-center in

Mask margin in

Clearance margin in

Silk screen outline

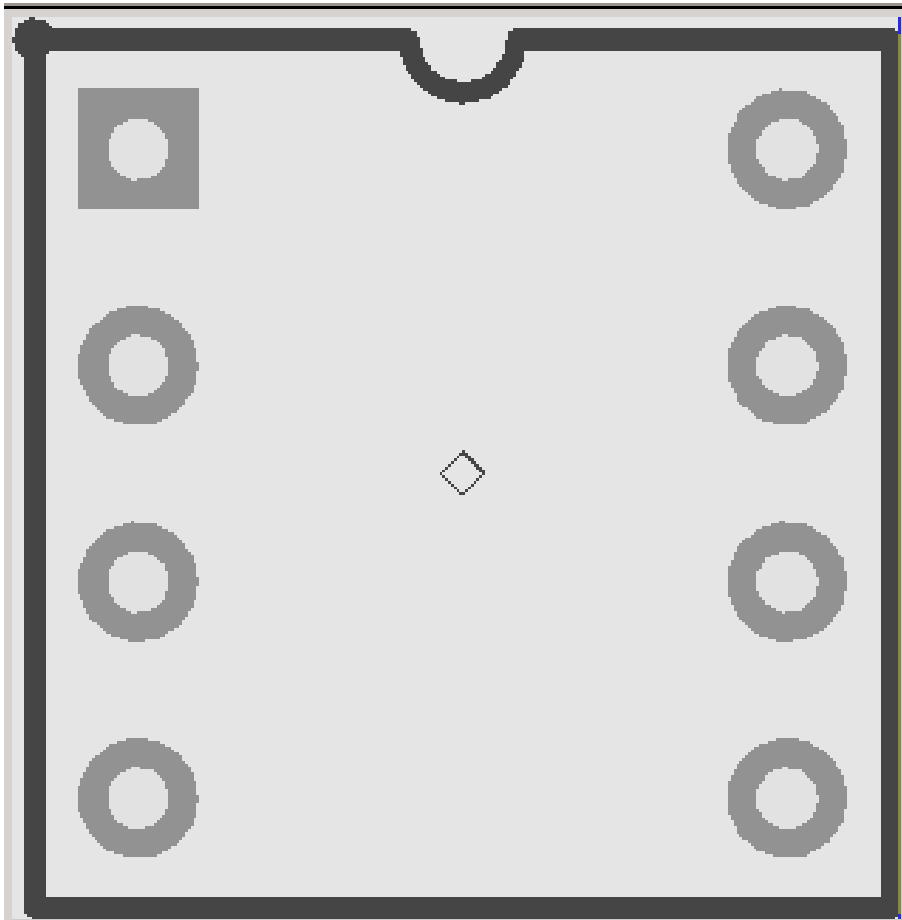
Pin 1 Dot

Quick Converter

mm = inch

- Create footprints for DIPs, connectors / headers and Surface Mount parts (resistors, caps, QFP, TSSOP, etc)
- Specify pad type, pad length and width
- Specify number of pads
- Specify center-center specs

Footprint Wizard



- For silk screen you can select the outline option or the pin 1 dot option. Both are enabled here.
- Pin 1 also gets a square pad.

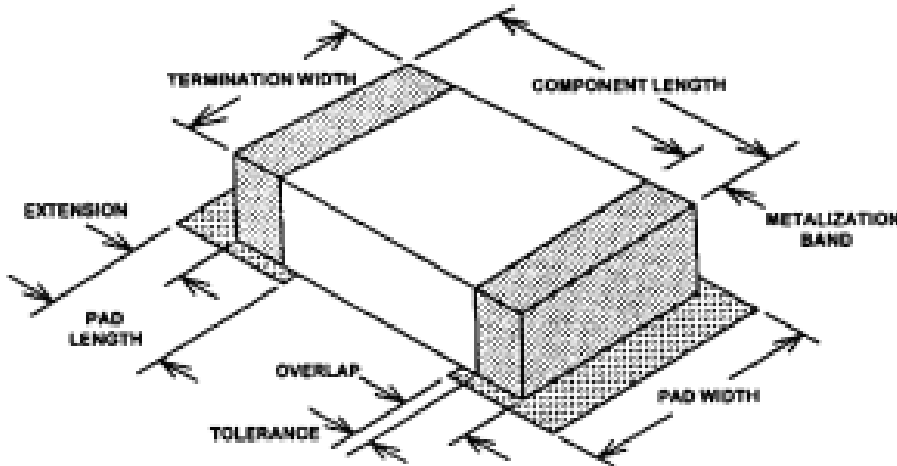
Pad Dimensions

Pad width = termination width + 0.010

Pad length = termination band + overlap + extension

Left-Right center – center = body length + pad length - 2 * (overlap + term band)

Pad spacing = body length - 2(termination band + overlap)



Pad Calculator for Resistors, Caps

Pad Calculator

For resistors, capacitors (in inches)

0.080	Body Length
0.050	Termination Width
0.015	Termination Band Width
0.020	Extension
0.002	Overlap

0.06	Pad Width
0.037	Pad Length
0.046	Pad Spacing
0.083	Center to Center

Load Footprint Wiz Cancel

Example is for an "0805" package

- Specify component dimensions
- Pad characteristics are computed
- Click button to transfer pad data to Footprint Wizard

Pad Dimensions

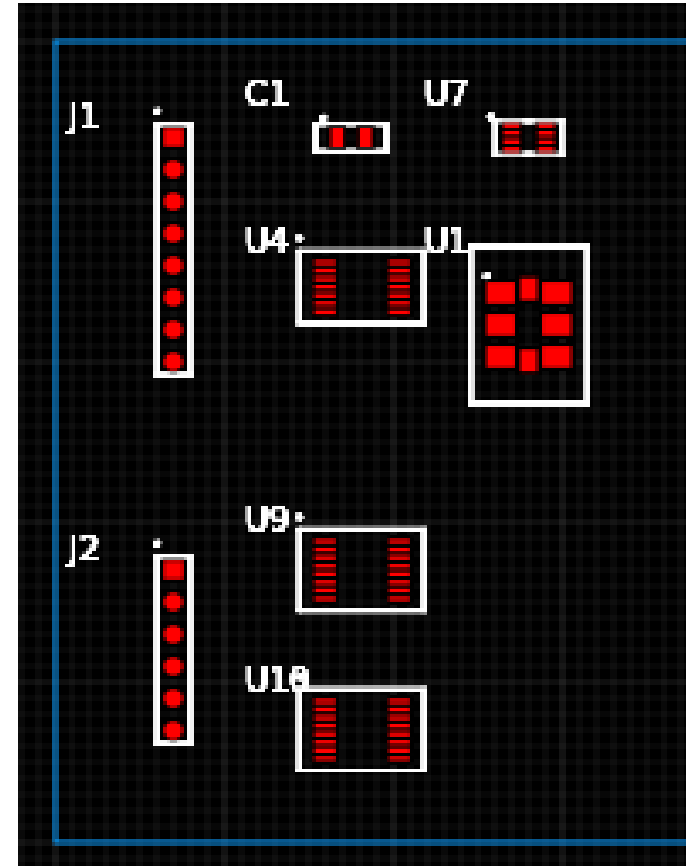
Device	Body Length	Term Width	Band Width	Pad Width	Pad Length	Left Right Ctr-Ctr	Pad Spacing
0201	0.020	0.010	0.005	0.015	0.016		0.008
0402	0.040	0.020	0.010	0.030	0.027		0.016
0603	0.060	0.030	0.012	0.040	0.034		0.032
0805	0.080	0.050	0.015	0.060	0.037		0.046
1206	0.120	0.060	0.020	0.070	0.047		0.076

Footprint Concerns

- There are many, many different packages in the world
- Circuit board manufacturers have similar but different guidelines and specifications
- SMT Pads should not be greater than 0.010 larger than the recommended sizes
- It is wise to try actual parts on a x1 printout of the artwork to verify that they are a match

Placing Parts

- Use menu **Comp > Unplaced Parts** to select and place location critical parts like connectors
- Use **Comp > Autoplace** a few times to place remaining parts
- Adjust locations to make room for vias and routes



J1 and J2 manually placed. Autoplace loaded some parts. Move them for spacing and no overlap. Repeat until all parts are placed. (Note: it turns out swapping U1 and U4 gives a better solution.)

Linking Netlist to Footprints

- Use menu ***Design > Update*** to update pin connection data to the loaded netlist

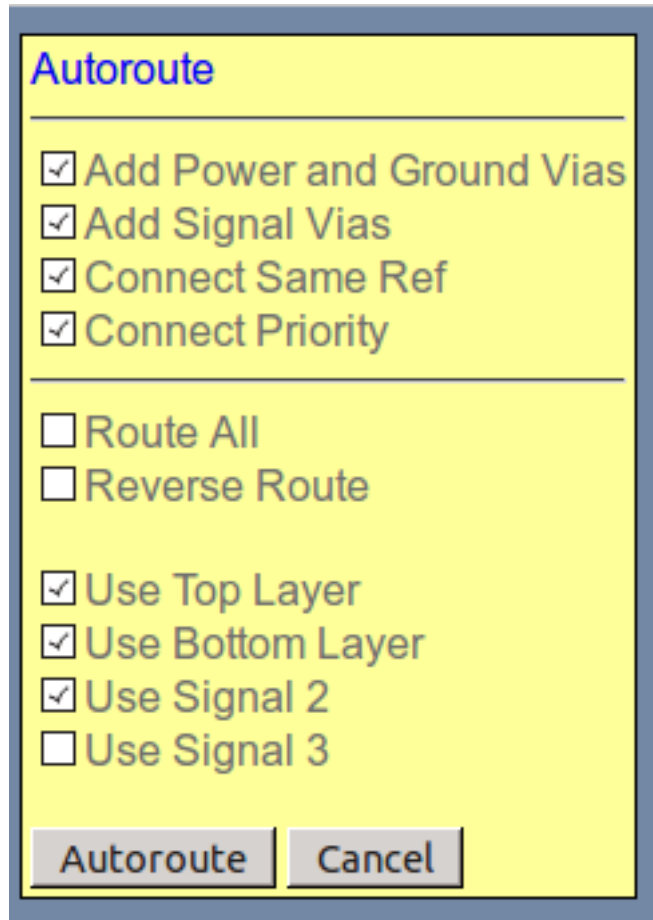
Route

- Use menu **Comp > Autoroute**
 - Note Autoroute is in development and probably will route only part of the design.
- Manually route remaining interconnects
 - Select a pin and from the pin menu use the following
 - **Add Net** – to just add a net
 - **Connect To..** - to select a destination and help you manually route
 - **Route To..** – to Autoroute to selected destination
 - **Route Net** – to Autoroute the entire net
- Use **Comp > Highlight Net** to highlight an entire net
 - Use **Edit > Delete** to remove an entire highlighted net

Autoroute is in Development

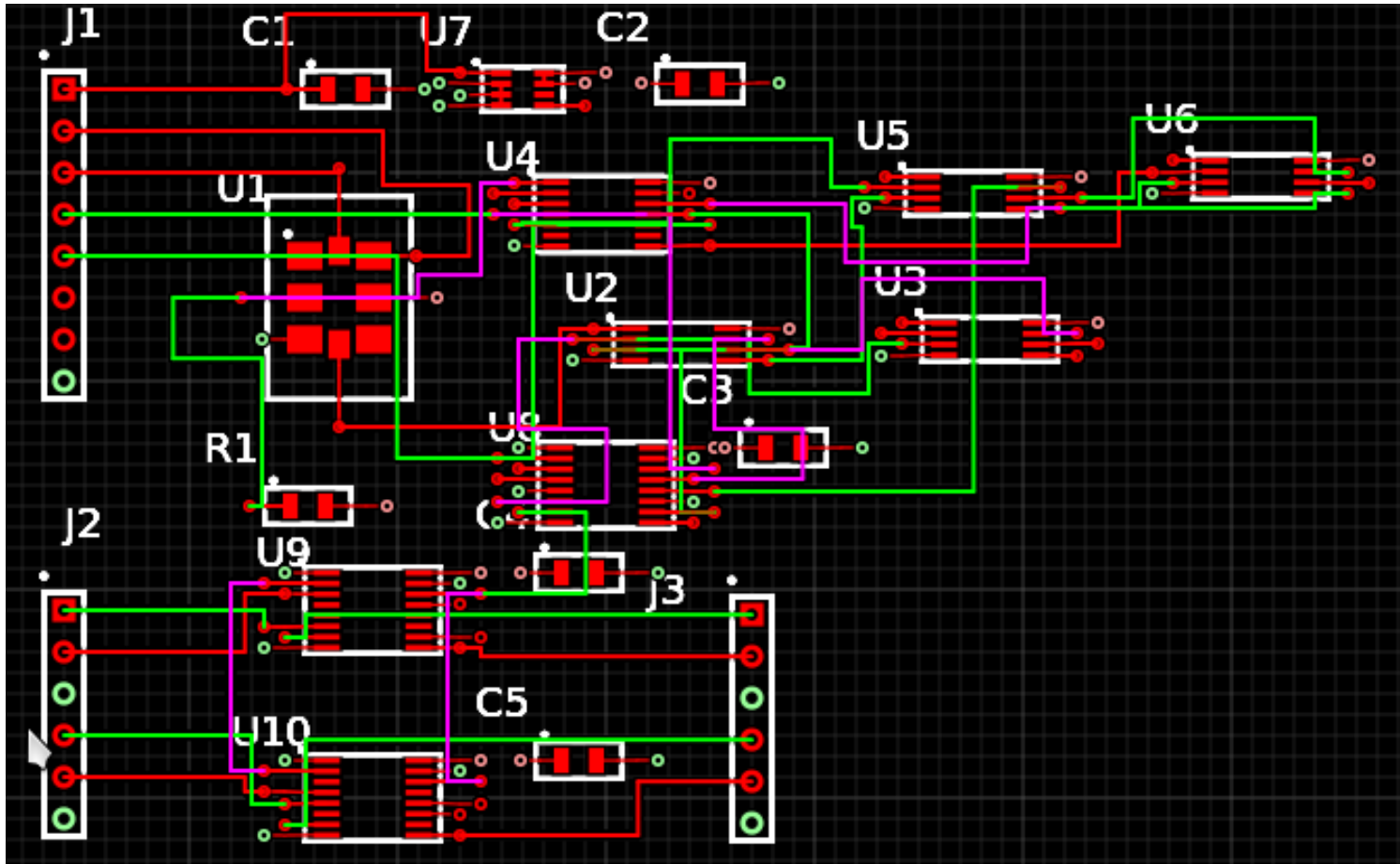
- Comp > Autoroute currently is limited
 - Adds connection and vias to power and ground
 - Connects most same-part connections
 - Attempts connections on the selected layers
- Autoroute takes time to complete (a few minutes +).
Be patient.
 - Your browser may prompt you that it's taking too long, but click to continue
- Use Comp > Remove All Nets/Vias to “start again”

Autoroute



- Autoroute takes time to execute. Doing it in a few passes lets you inspect intermediate results.
- On first pass, select the top four options to add vias and connect critical signals
- On second pass, only select the Route All and Reverse Route options

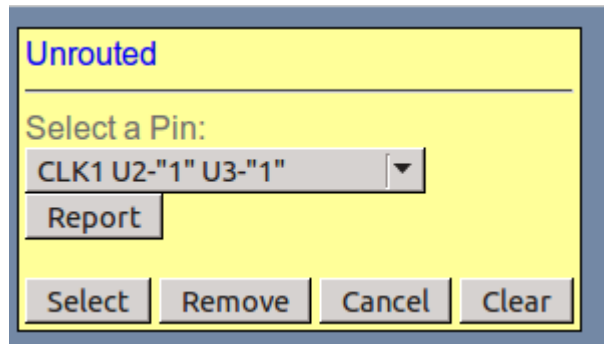
Autoroute Solution



Unrouted Segments Are Reported

Unrouted Report 4/14/16

CLK1 U2-"1" U3-"1"
 CLK2 U4-"11" U8-"10"
 CLK2 U2-"3" U3-"2"
 CLK2 U3-"2" U6-"1"
 CLK3 U6-"3" U6-"5"
 CLK4 U5-"1" U5-"6"
 CLK4 U4-"3" U8-"4"
 CLK4 U4-"3" U5-"1"
 CLK5 U3-"5" U5-"3"
 CLK5 U2-"5" U4-"2"
 CLK6 U2-"7" U3-"6"
 IIN_N U9-"3" U9-"10"
 IIN_P U9-"6" U9-"13"
 LOGIC_HIGH U4-"1" U4-"13"
 N_00000048 U2-"6" U8-"3"
 N_00000054 U5-"7" U6-"6"
 PHASEQ U8-"9" U9-"2"
 QIN_N U10-"4" U10-"11"
 QIN_P U10-"5" U10-"12"
 VPWR U7-"5" U7-"1"
 VPWR C1-"1" U7-"5"



- Unrouted net segments were not successfully processed by Autoroute
- Selecting each from Comp > Unrouted menu invokes the “Connect To” function

ESC clears the Unrouted list

About Traces

Trace Width	Current Capability
0.010 inches	0.3 Amps
0.015	0.4
0.020	0.7
0.025	1.0
0.050	2.0
0.100	4.0
0.150	6.0

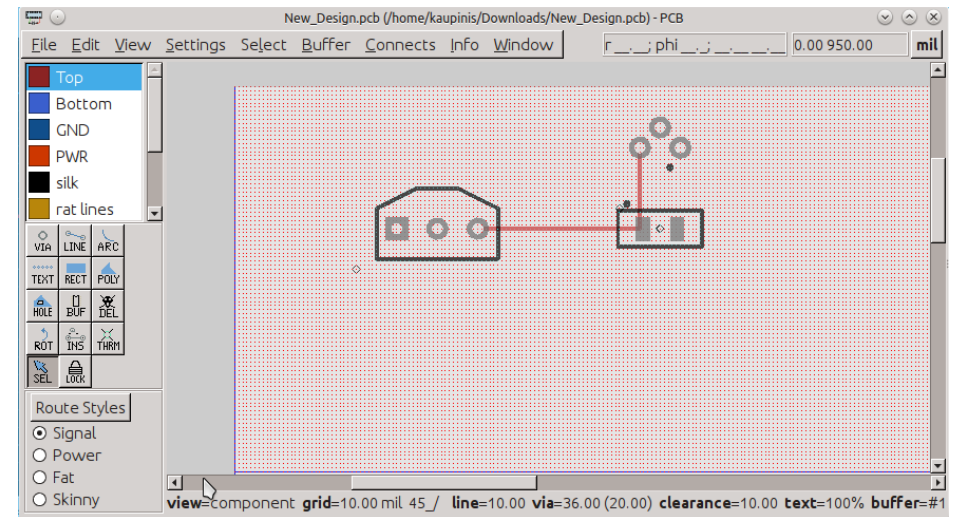
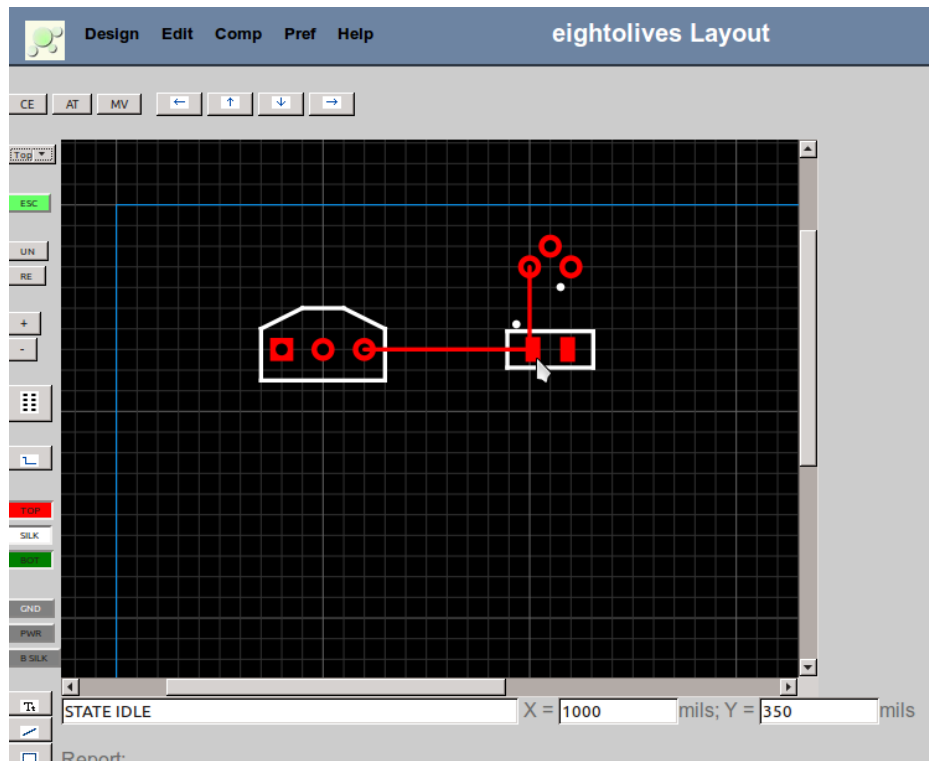
For example, for pads spaced 0.025 in, a trace width of 0.010 with a gap of 0.015 provides some margin.

- Minimum gap between traces is mfg dependent
 - 0.007 min
 - 0.010 recommended
- Minimum width is mfg dependent
 - 0.006 min
 - 0.010 recommended
- Use eightolives' Microstrip Quick App to estimate trace impedance

Save Your Work Often

- Circuit layouts can be complicated. Save your work at various stages to simplify “going back”
- ***Design > Save Project***
- Recommend saving (with different file names)
 - The empty board outline
 - The outline with critical parts placed
 - The final placement
 - Final placement with critical routes
 - Various final with additional routes
 - Final design

The .pcb file output is compatible with gEDA's PCB tool



.pcb file opened with gEDA's PCB which can also be used for changes, checking routing and artwork

Design created with Layout .
Saved as a .pcb file

About Compatibility

- The primary goals of compatibility with gEDA PCB are:
 - PCB can read .pcb files created by Layout
 - Layout can read a subset of PCB constructs (limited only by priority and development resources)

Non-compatible Constructs

- Layout can also use non-compatible constructs to address growth situations not handled by gEDA PCB
- `#NON-COMPATIBLE_CONSTRUCTS` flag warns of such constructs
- Non-compatible constructs include:
 - Milling/Router holes on board and in footprints
 - Adding Lines, Polygons to footprints
 - Micro-vias

Slots



- A slot is an oblong cutout shape that is either created using a milling machine or by drilling closely spaced holes along a straight line.
 - Its specification is not defined by either gEDA PCB or RS-274X version 2.
- Layout addresses the issue in 2 ways
 - A footprint Pad with the “hole” flag set is considered a slot. The graphic will appear on the Drill Map.
 - A Net drawn on the Outline layer defines a slot

Artwork Layers

Top Silkscreen
Top Solder Paste
Top Soldermask
Top Copper
Ground Copper
Signal 2 Copper
Signal 3 Copper
Signal 4 Copper
Signal 5 Copper
Power Copper
Bottom Copper
Bottom Soldermask
Bottom Solder Paste
Bottom Silkscreen

Profile (Board Outline)
Drill Map
Plated-through Drills
Unplated Drills
Rout / Milling

- 1 layer boards can be either Top for surface mount or Bottom for through-hole only.
- 2 layer boards are Top and Bottom
- The Ground and Power planes are exclusive (no other signals allowed)

Design > Generate Artwork

Generate Artwork Files

Design Name

Select Files:

- Layer Top
- Layer Bottom
- Top Soldermask
- Bottom Soldermask
- Layer GND
- Layer PWR
- Layer Silk
- Layer BottomSilk
- Layer Outline
- Layer Signal 2
- Layer Signal 3
- Drills (.gbr)
- Drills (.xln)
- README.txt

- Note: This code is in development and is not complete
- Select files to be created.
- Files are saved in the selected Project area (LOCAL is default)
- Check the files using third-party tools such as gerbv

For more information

- Visit <http://www.eightolives.com>
- Board Design Guidelines (Altron Inc)
 - <http://www.altronmfg.com/pcb-design-for-manufacturability/>
- The Gerber Format Specification
 - Ucamco Revision 2015.10
- For the gEDA Project Wiki visit <http://wiki.geda-project.org>