

gEDA

Larry Doolittle



Open Hardware Workshop, Grenoble 2011

Outline

- 1 Philosophy
- 2 Mechanics (PCB flow)
 - Component representation/Part creation
 - Schematic Entry
 - Layout
 - Manufacture
- 3 Notes and Summary

Philosophy of Pluralism and Constructivism

Unix heritage shows

- Constructivist mentality: GUI programs are seen as components of the EDA suite, not the home one lives in
- File formats as simple as possible to encourage reading and writing by users and other software
- Encourages clear separation between source and derived files, good for checking source files into a revision control system
- Tools provide capability, not set policy
- Example: even GUI tools provide command-line configuration and control to perform actions like producing printout in a script-able and Makefile-compatible way

Elements of gEDA

- **gschem**: schematic editor
 - **pcb**: PCB layout
 - **library**: for schematic and layout
 - **gerbv**: Gerber RS-274X file viewer
 - **Icarus**: Verilog simulator (some synthesis capability)
 - **gtkwave**: wave viewer
- + many smaller utilities and community projects

Specialization for PCB layout

(but many of the tools used are not restricted to this flow)

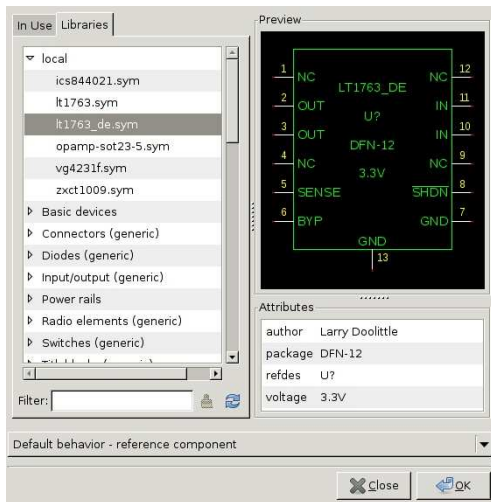
- Data-centric view; at its most abstract consists of
 - Component list, with various attributes: package, footprint, value, orderable part number, supplier, cost, tolerance, material
 - Net list: establishes connectivity
- Schematic is seen as one tool to create and edit the data
- Layout tool's most important job is to check the artwork for consistency with the data

PCB flow

- Part management and creation
 - this talk shows `djboxsym` for symbols and `footgen` for footprints
- Schematic entry with `gschem`
- Layout with `pcb`
- Gerber viewing with `gerbv`
- Manufacturing
 - Mailing list has discussions of DIY photographic and milling techniques
 - Of course normally designers send Gerber etc. files “out for fab”

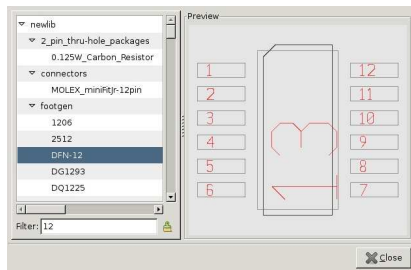
djboxsym by DJ Delorie

```
[labels]
LT1763_DE
refdes=U?
package=DFN-12
voltage=3.3V
! author=Larry Doolittle
[left]
1 NC
2 OUT
3 OUT
4 NC
5 SENSE
6 BYP
[right]
12 NC
11 IN
10 IN
9 NC
8 SHDN
7 GND
[bottom]
13 GND
```

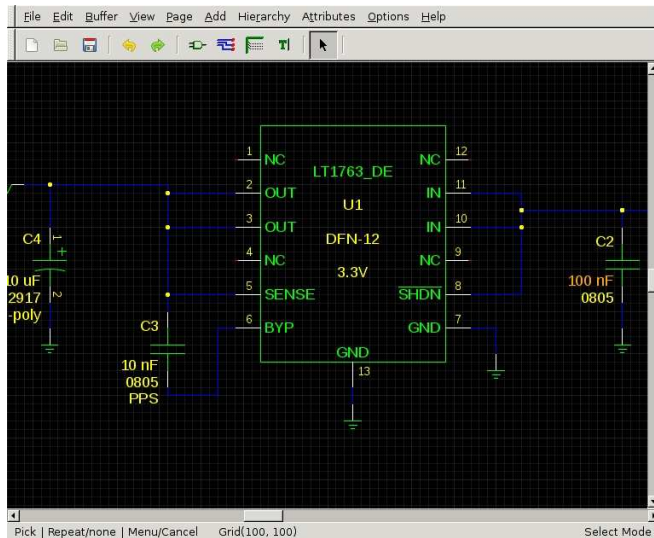


footgen by Darrell Harmon

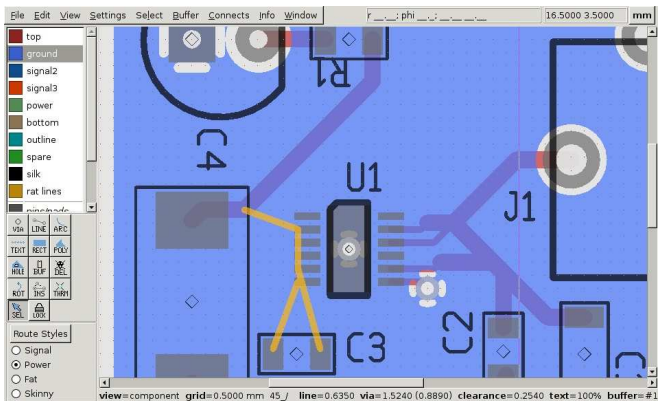
```
# LT1763CDE 4x3mm DFN
type "so"
pitch = 0.50 mm
pins = 12
padwidth = 1.0 mm
padheight = 0.3 mm
width = 2.2 mm
ep = 1.7 mm
rect = 3.3 mm
silkboxheight = 3.1 mm
part "DFN-12"
```



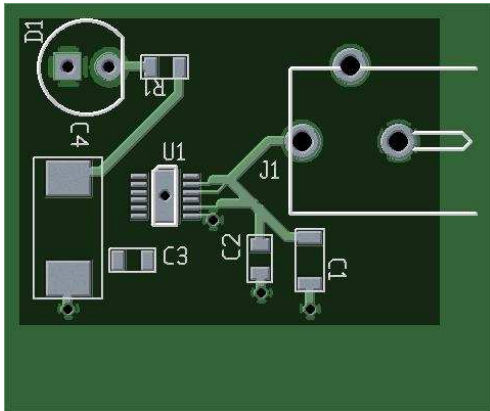
gschem screen shot



pcb GTK screen shot



pcb photo mode preview



pcb capabilities

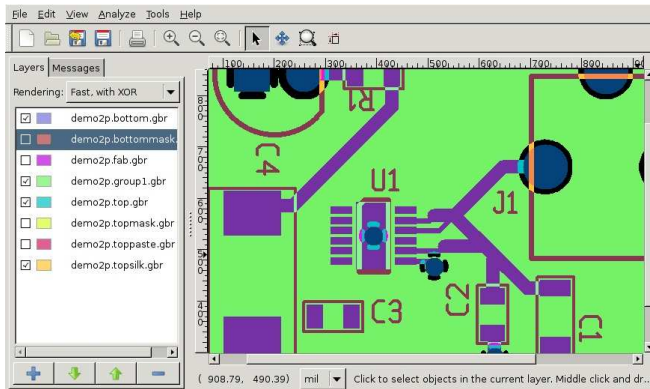
Usability

- Configuration (board size, layers, design rules)
- Netlist import, or full design import/update from `gschem`
- Component placement
- Wiring and part movement
 - mouse, /, and digit keys for wiring
 - careful use of grid increases quality of results
- Wiring check "Optimize rats nest"
- Design Rule Check "DRC Check"
- Export and print Photo-preview
- Export Gerber and other manufacturing files

Limits

- pad stacks are only minimally customizable – no blind vias
- footprints consist only of pins, pads, and silk – no escape routing
- standard autorouter isn't very good, topological autorouter is incomplete
- no IPC-D-356A netlist export yet

gerbv screen shot



Resources

Starting point:

- <http://gpleda.org/>

Tutorials abound, some quite obsolete

- <http://geda.seul.org/wiki/geda:documentation>
- http://geda.seul.org/wiki/geda:gsch2pcb_tutorial
- <http://www.delorie.com/pcb/docs/gs/g.html>

Versions

- gEDA/gaf 1.6.2-20110115 and pcb-20110918 used here
- Linux distributions usually have older versions this software pre-packaged. It's easy to recompile a newer version from source if needed.
- Bleeding edge sources are available via git, only recommended if you want to take an active role in fixing bugs or adding features.

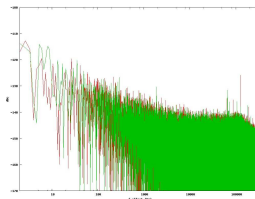
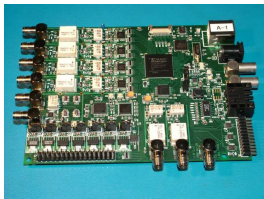
Platform

Microsoft Windows ports/builds of these programs exist, but ..

- are often reported to be half-broken
- miss the context of easy command-line scripting, Makefiles, etc.
- could be useful in some contexts, but give a poor first impression
- recommend using the Unix (Linux, or maybe Mac) versions at first to see full capabilities

Summary

- The gEDA umbrella includes a collection of projects that combined, enable hardware design using only Free Software
- The software has evolved over a number of years, and continues to evolve, staying consistent with a modular toolkit
- Even though there is always room to improve, the suite is complete and has been used for many successful designs



Merci Beaucoup!