Using gEDA

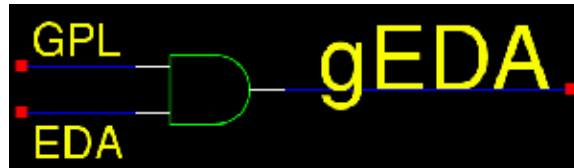**by Iznogood**

<iznogood/at/iznogood−factory.org>

About the author:

Involved in GNU/Linux for a while, I'm now running a Debian system. Despite electronic studies, I've mostly done a translation work for the GNU/Linux community.

Abstract:

How to develop some Printed Board Circuits with free tools from the gEDA community.

Introduction

Why another article to talk about EDA, Electronic Design Automation? Because there is more than one way to do it and this one is free. One of its best particularity is the fact that we have a set of tools, some graphical ones but mostly command line based. With this article, I will just show how to make a PCB (Printed Circuit Board) from the beginning.

What is gEDA?

Most important, gEDA is a set of free (in the GPL way) electronic tools. It is therefore always in development state. I put here the gEDA diagram. Click on it to enlarge.

gEDA Overview
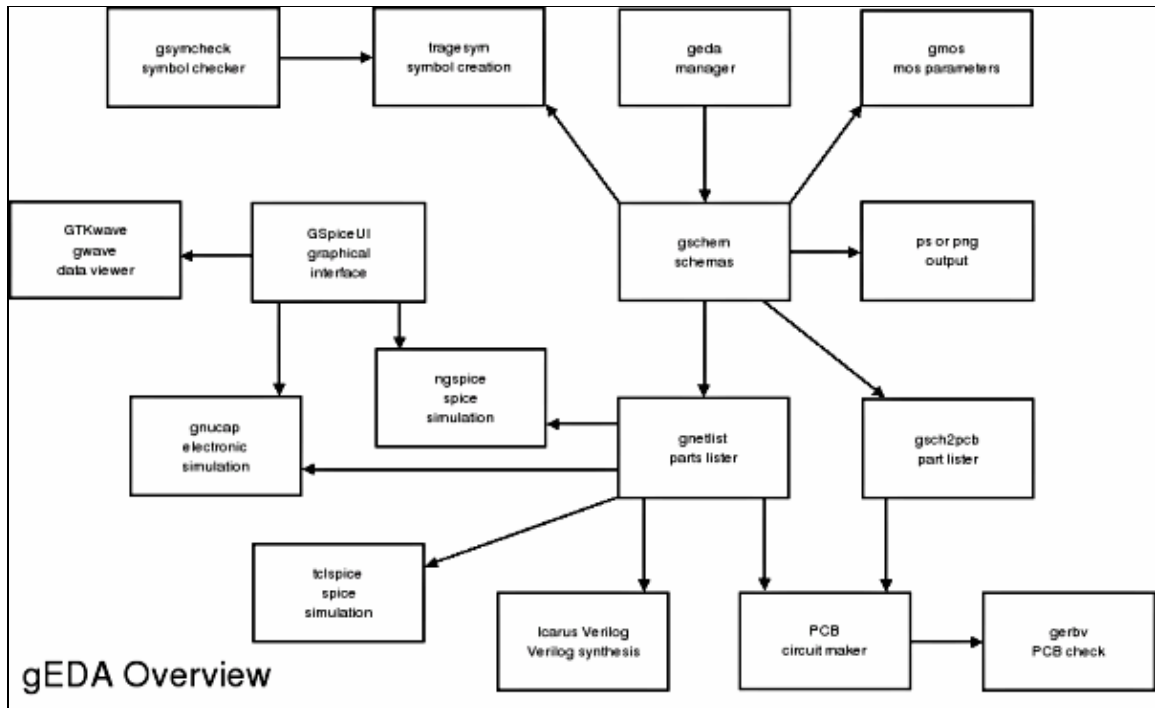
In the center, we have gschem, the graphical schematic and symbol editor. It can be called by geda, the manager, on top of gschem. With it, you can not only launch gschem but a spice simulator, a pcb designer, a gerber and a waveform viewer.

On the top left, with tragesym, you can create symbols and gsymcheck will check them.

Most interesting for us, down gschem, we have two ways of doing net−lists. It is a gEDA particularity: every exchange file is in plain text, only the extension indicate the goal:

- .sym for symbols files
- .sch for schematic files
- .net for net−list files
- .pcb for pcb files and so on...

You have gnetlist, the "official" way to do net−listing for spice simulation with tclspice, ngspice, gnucap, Verilog synthesis with Icarus or PCB creation. gsch2pcb is a net−lister dedicated for pcb. It does the translation from .sch file components to pcb file, back and forth. And to finish, we check the pcb file with gerbv.

In this article I will only talk about gschem, gsch2pcb and PCB. We will begin with gschem.

Placing components with gschem

I won't talk a lot about installation: apt−get install geda−gschem geda−symbols geda−utils pcb for GNU/Debian.

On others systems, download the required files on http://geda.seul.org/source.html, http://pcb.sourceforge.net/ for PCB and ./configure & make & make install as usual.

Ok, let's open gschem. We have two ways to do it: we open it with the command gschem alone or we put a filename like: *gschem sample.sch*. Now we can place components: one more time, we have two choices: with the mouse and "Add Component" or from the keyboard with "i". We select BNC−1.sym, pwrjack−1.sym in connector, 2 resistor−2.sym, 1 resistor−variable−2.sym, 2 capacitor−1.sym, 2 capacitor−2.sym, bridge−1.sym

in analog, lm317−1.sym in linear. We will have this picture as result:
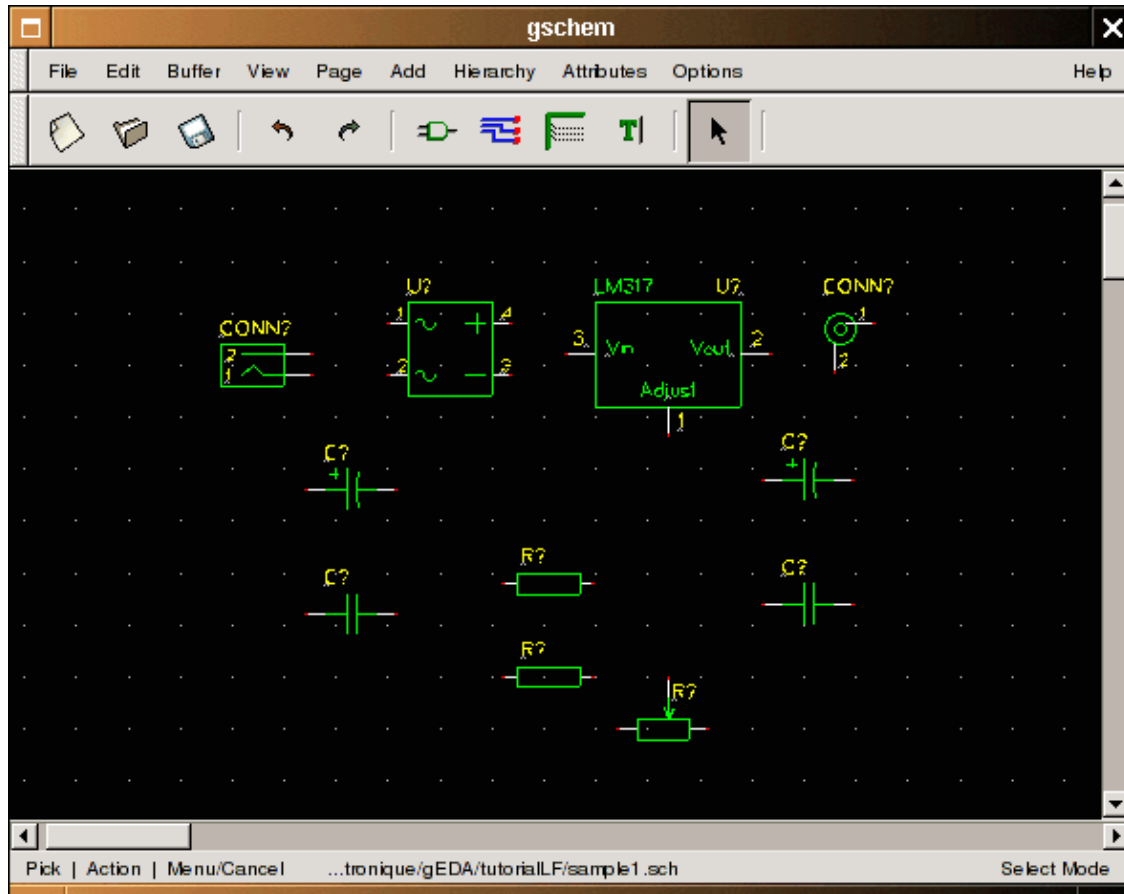
Don't worry, you can download all files from my homepage (updated) or from this server.
Now we can wire the components. To do it, we need to move them and make rotations. Since we are lazy people, we will use keyboard shortcuts: click on component then "er" for rotation, "ei" for mirroring, "m" for moving, and "n" for wiring the components. We can select only the part name to move or rotate it.
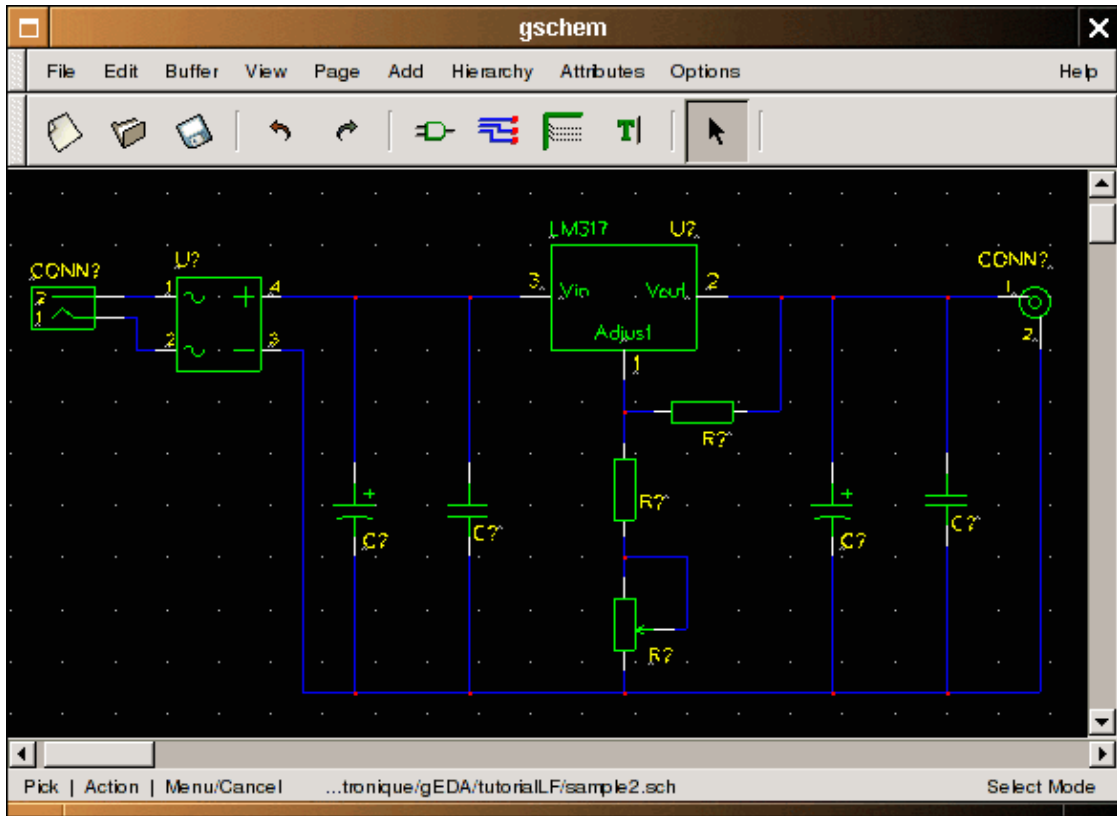
Once done, we can edit each component attribute with "ee". We can begin with the most important component : LM317. When we open the attribute box, we can see this:
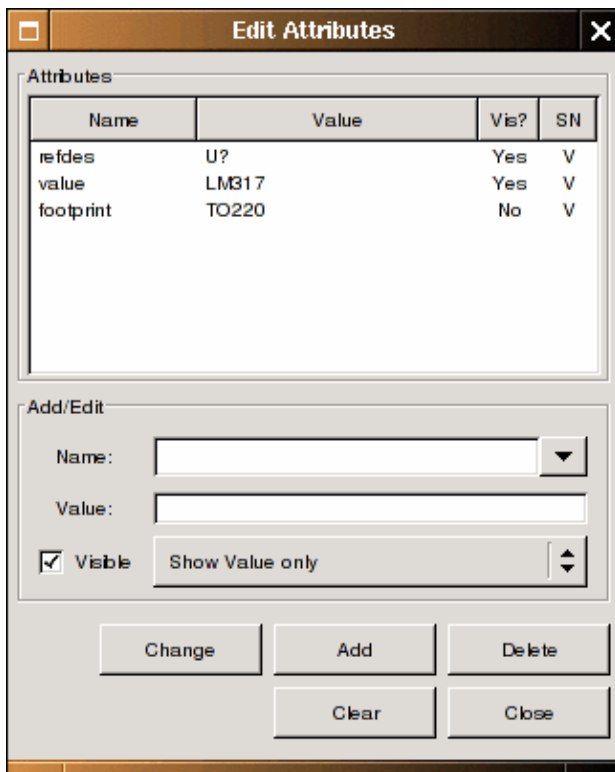
We write LM317 into value name and TO220L for footprint (make it invisible). For resistors, the value is indicated above on the schema and the footprint is R025. And so on for capacitors, potentiometers (you can see the sample.sch file added to the article)... Once we have finished, we can save it with a "fs" if the file have a name and "fa" otherwise.

Translation with gsch2pcb

First, we need to give a number to the parts with the command:

```
refdes_renum sample.sch
```

If we reopen the file, we will see all ? from R?, C? U? changed to a number. This mean we can make a net−list. First we must install some footprint library and make a project file. We need to do it because there is a lack of documentation for pcb footprint use with gschem or I haven't found it. A good footprint set can be found here : http:// web.wt.net/%7Ebillw/gsch2pcb/gsch2pcb−libs.tar.gz. (local copy)
Download and install it with:

```
mv gsch2pcb-libs.tar.gz ~/toTheLocationYouWant/
tar xvzf gsch2pcb-libs.tar.gz
```

Then you can make a project file with:

schematics sample.sch
elements−dir ~/XXX/pcb−elements
output−name sample

I have named it sampleLFprojet but you can use the name you want. The schematics is for our gschem file. Elements−dir gives to pcb the footprint directory to use (replace XXX with your directory location for pcb−elements). And the output−name is to continue with the sample name for pcb. We save it in the same sample.sch directory.

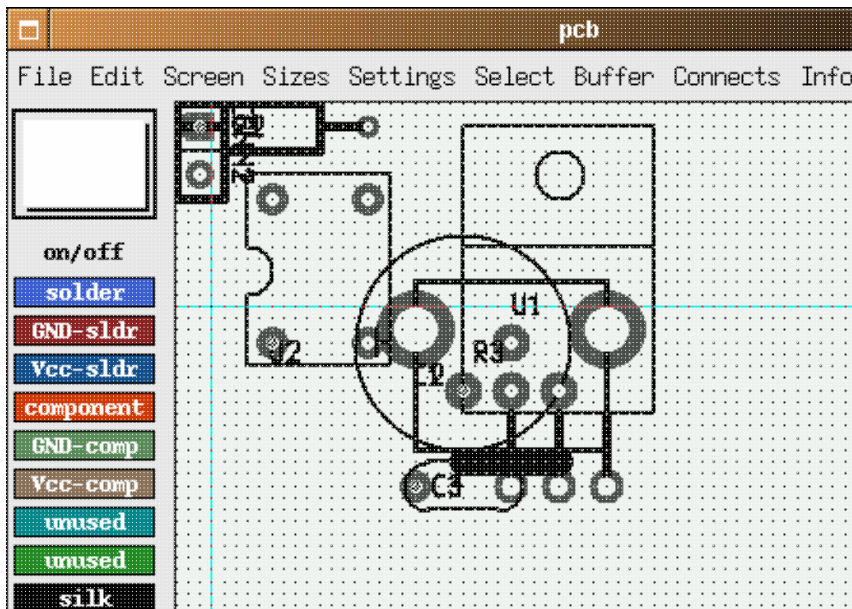The next step is to run:

```
gsch2pcb tutorialLFprojet
```

The output are files named sample.board and sample.net. The first one lists all component parts to be used with PCB and the second one is the net listing.
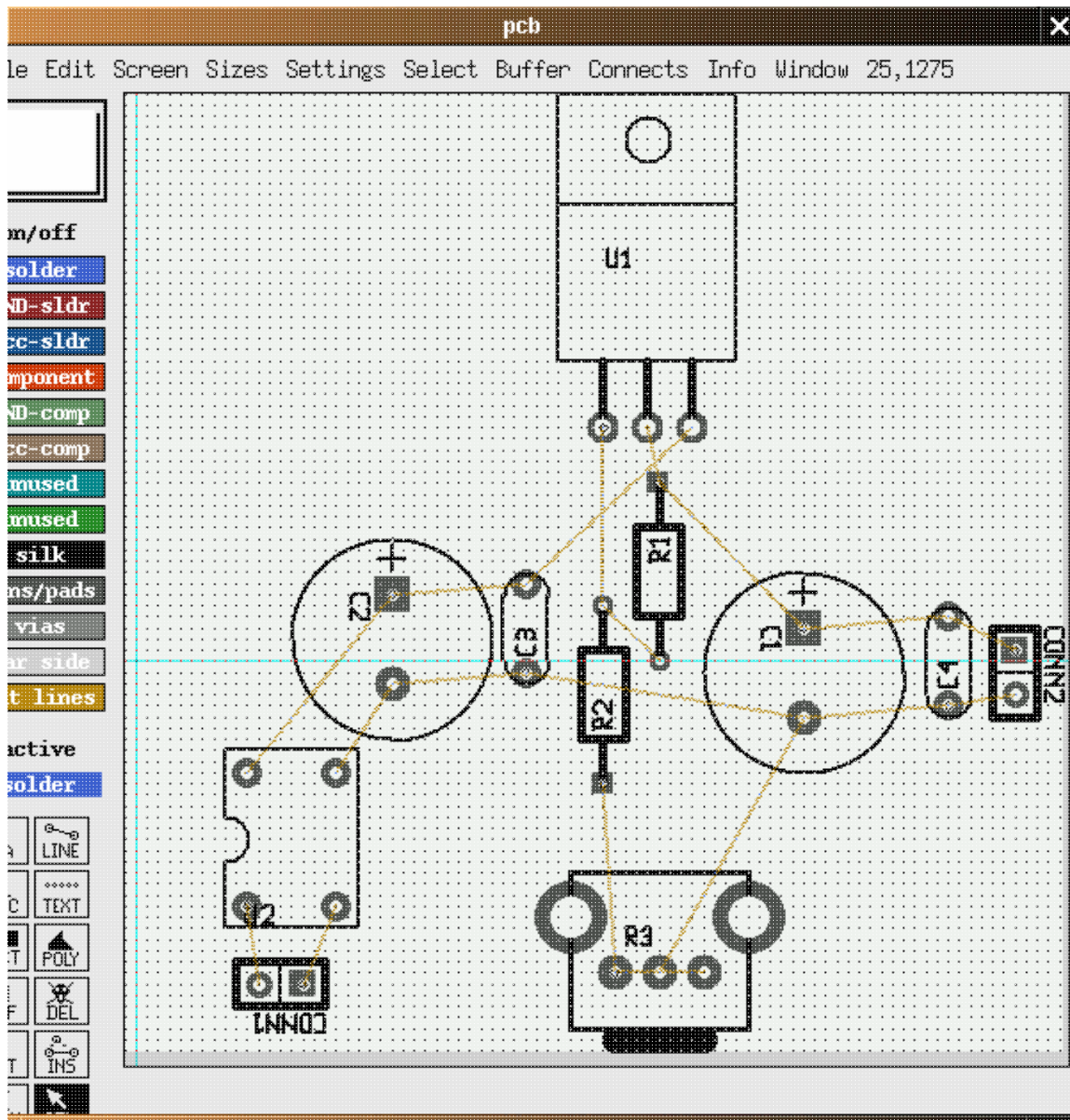
Making a circuit with PCB

We can start now with PCB. Open :
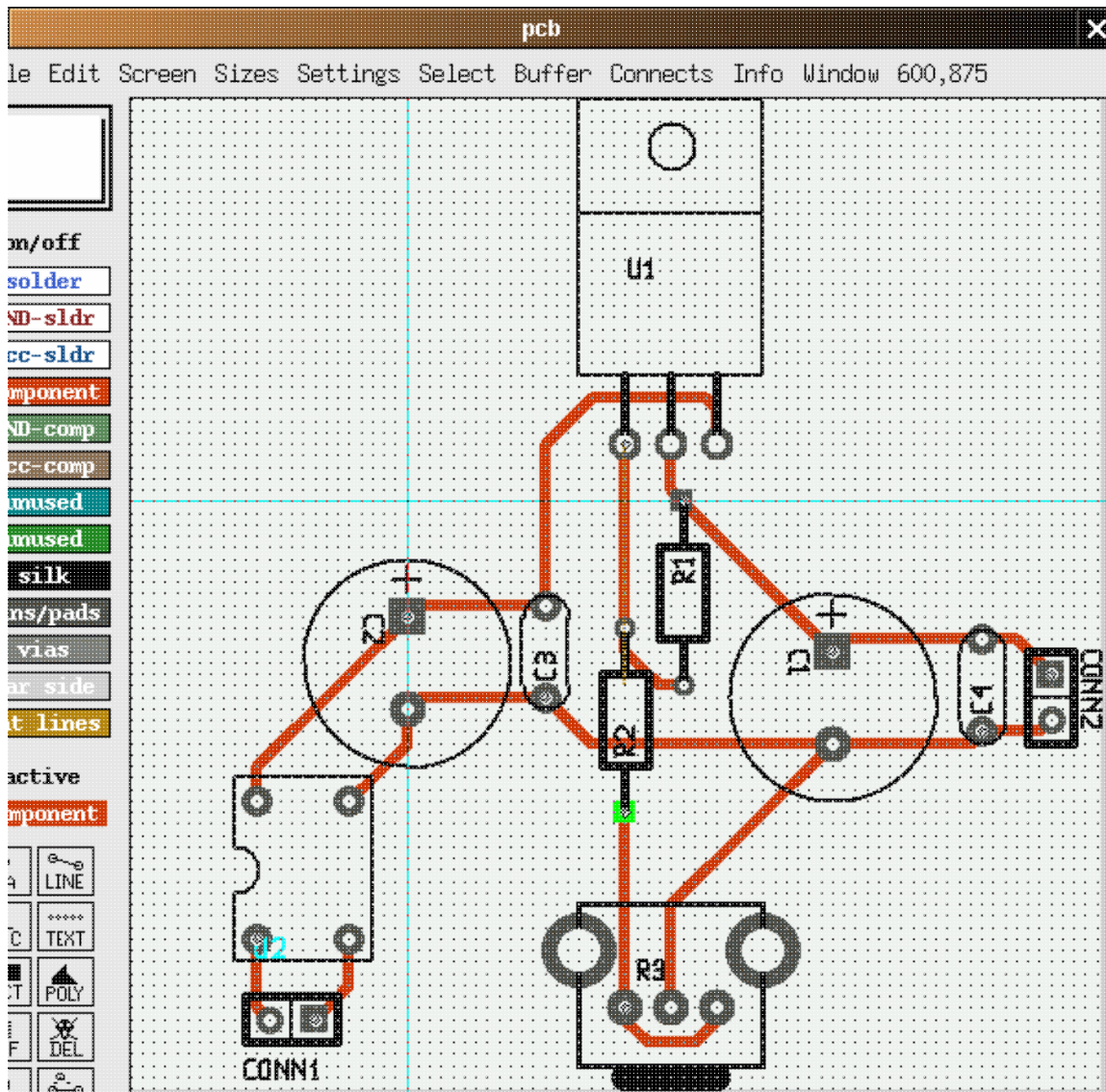
```
pcb sample.pcb
```

You will have this picture :

All components are packed to the left up side. We need to place the components on the board. Then we call the wires (nets) with the sample.net file with File −−> load net−list file and optimize Connect −−> rast−nets. It's quite magic.

Now we place nets as desired with Line and solder mask selection.

It's finished: you just have to save and make a File ––> print layout to have your pcb printed.

Conclusion

The goal of this article was just to have a straight and rough introduction to of some of the gEDA tools available and to show an easy, short way to make a PCB. But gEDA is so much more that it will take you a long time to play around with it: you will need to make a lot of optimizations to make an output usable for professional pcb makers.

Related articles

This article is mainly influenced and based on the Bill Wilson bill––at––gkrellm.net documentation gsch2pcb.html.
And you can find more information at www.geda.seul.org.
PCB can be found at pcb.sourceforge.net.
If you understadn french then you can find more on http://www.iznogood–factory.org and for english

speaking people, there is some stuff here.

2005−01−15, generated by lfparser_pdf version 2.51