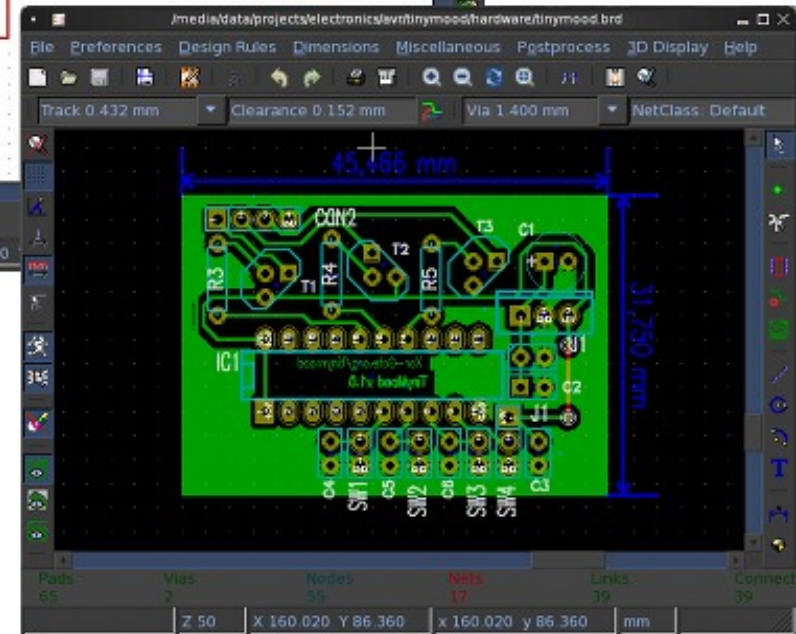
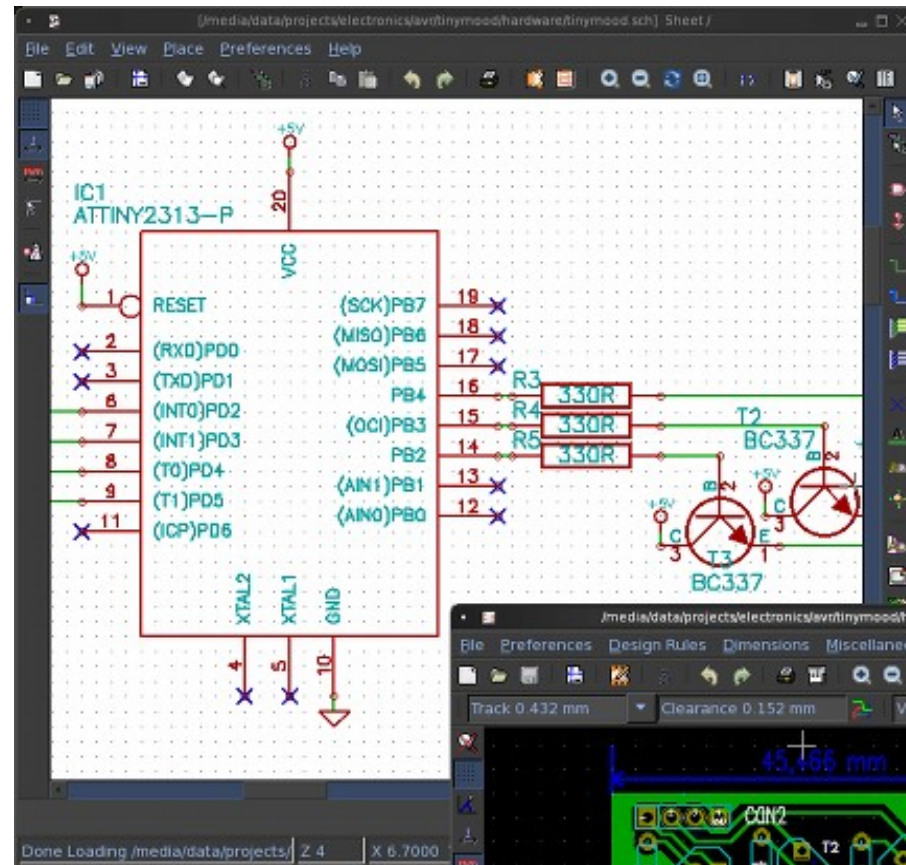


KiCad

The GPLed Pcb Suite

Jerry Jacobs
@ t-dose 2009



Introduction

- Who am I ?
- Why KiCad?
- Project cycle with KiCad.
- KiCad, project manager
- EESchema, schematic editor
- CvPCB, footprint assignment
- PCBNew, pcb editor
- GerbView, gerber viewer
- Professional Use!
- Short Demo

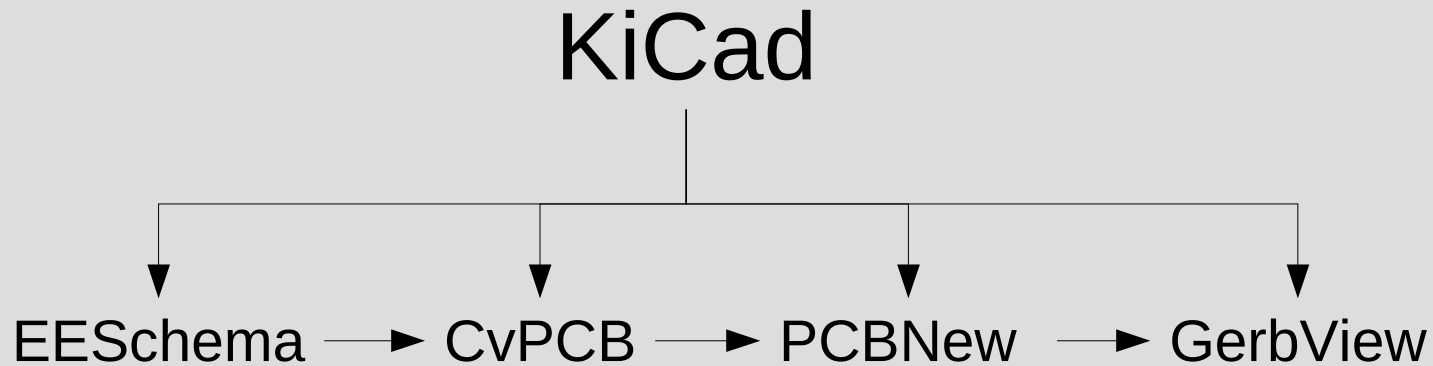
Who am I?

- Jerry Jacobs born in 1989
- An opensource hacker
- An embedded systems engineer (in progress)
- A linux and osx user

Why KiCad?

- Its open-source of course!
- Active development
- Active user group
- Complete written on wxWidgets toolkit
- Easy building using CMake
- Runs on Linux, *BSD, OSX, Windows
- Files formatted in ASCII format and system independent
- Complete documentation in OpenOffice

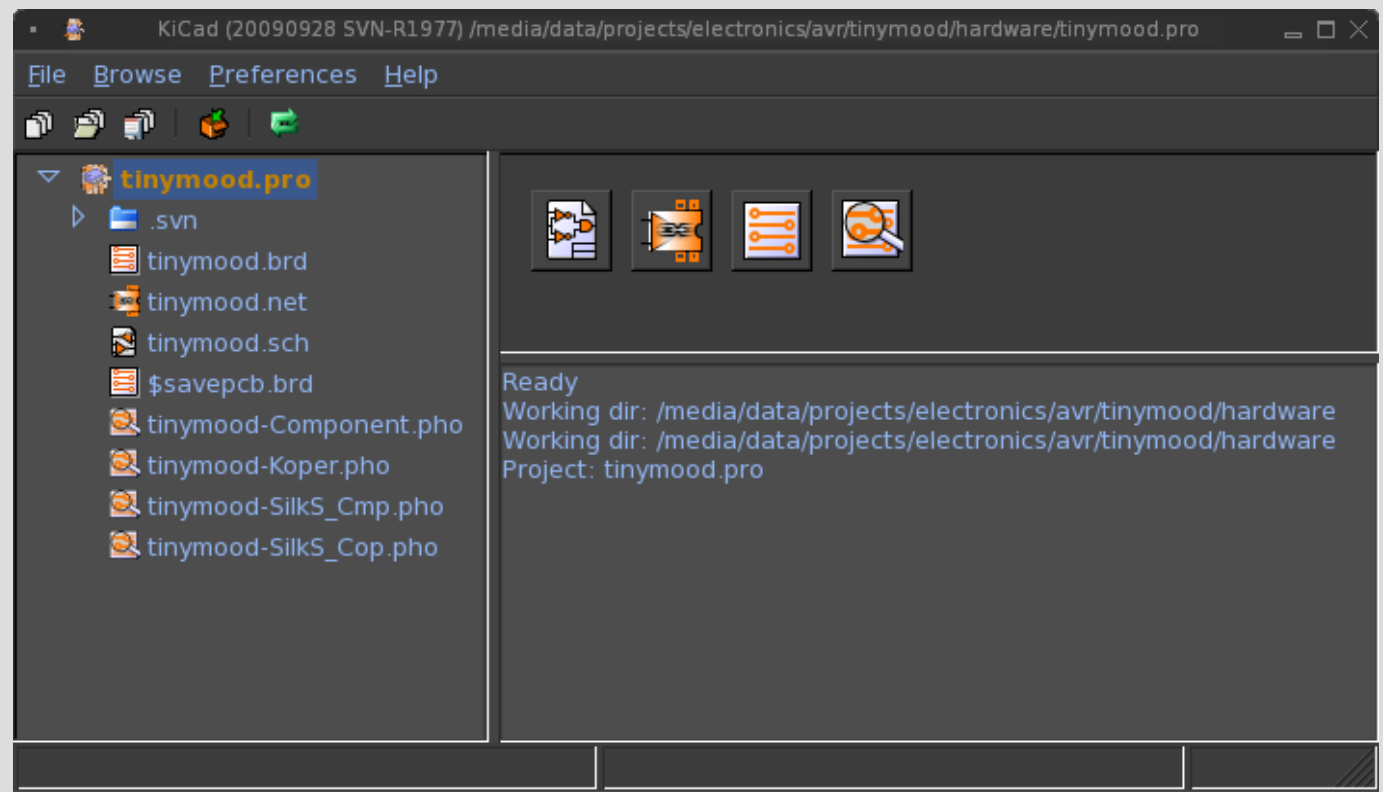
Project cycle with KiCad



KiCad	Project manager
EESchema	Schematic editor
CvPCB	Footprint assignment and viewer
PCBNew	Printed circuit board editor
GerbView	Gerber rs274x viewer

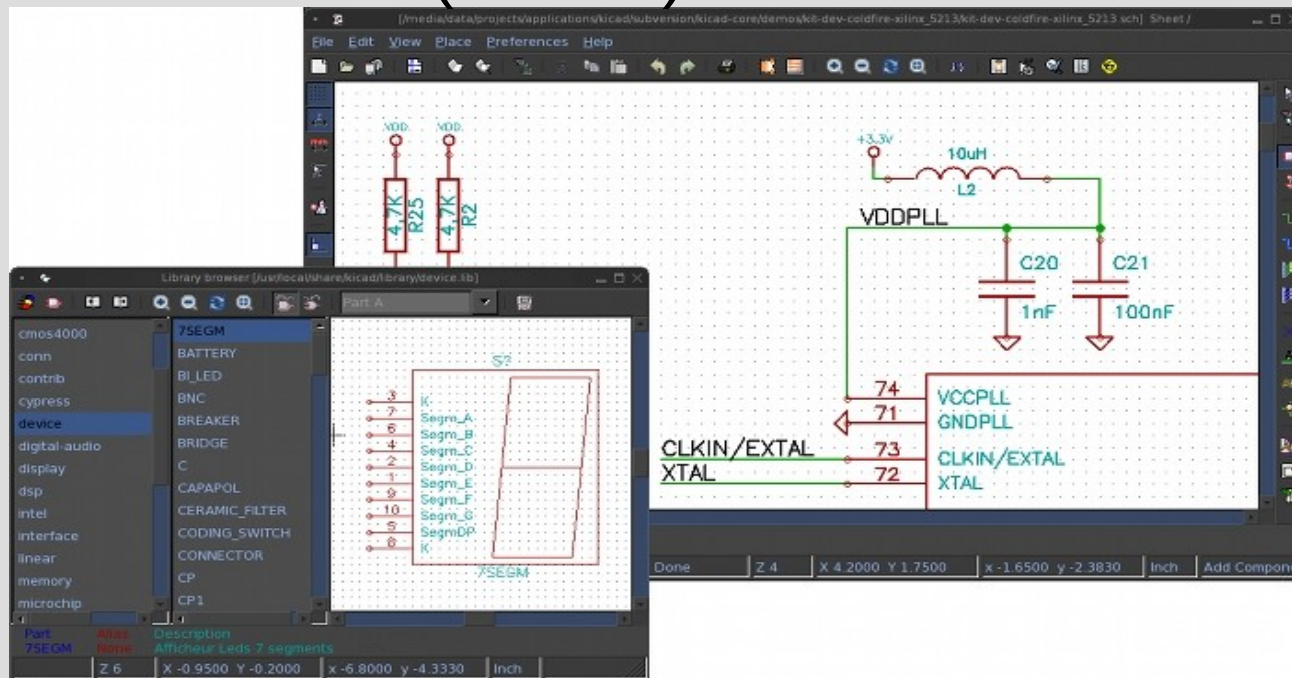
KiCad – Project Manager

- Glue for all the KiCad utilities
- Access all the programs in the pane
- Open file direct with associated program



EESchema – Schematic Editor

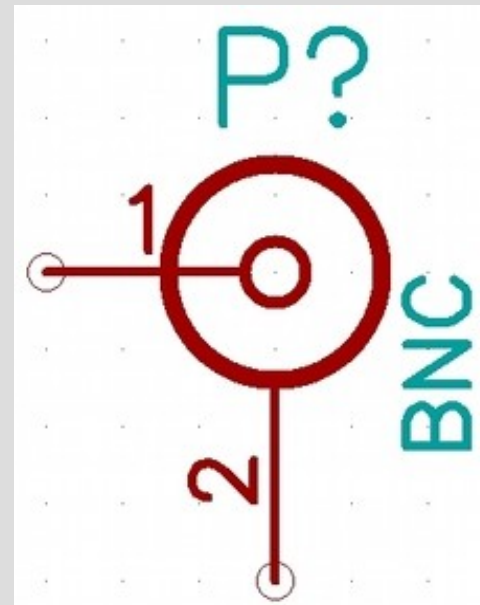
- Hierarchical drawings, using multi-sheet diagrams.
- Easy component creation, editing and visualisation.
- Handling of symbol libraries import, export, addition and deletion of components.
- Access CvPCB and PCBNew within EESchema
- Design rules check (D.R.C.)



EESchema – Schematic Editor

- Printing to Postscript, HPGL, SVG.
- Printing the schematic on a printer.
- Bill of Materials generation.
- Netlist generation for PCB layout or simulation software.
- The schematic and symbol library file format is well documented. (To write a Perl module against ?)

```
#  
# BNC  
#  
DEF BNC P 0 40 Y N 1 F N  
F0 "P" 10 120 60 H V C CNN  
F1 "BNC" 110 -60 40 V V C CNN  
DRAW  
C 0 0 70 0 1 12 N  
C 0 0 20 0 1 8 N  
X In 1 -150 0 130 R 40 40 1 1 P  
X Ext 2 0 -200 130 U 40 40 1 1 P  
ENDDRAW  
ENDDEF
```



CvPCB – Footprint Assignment

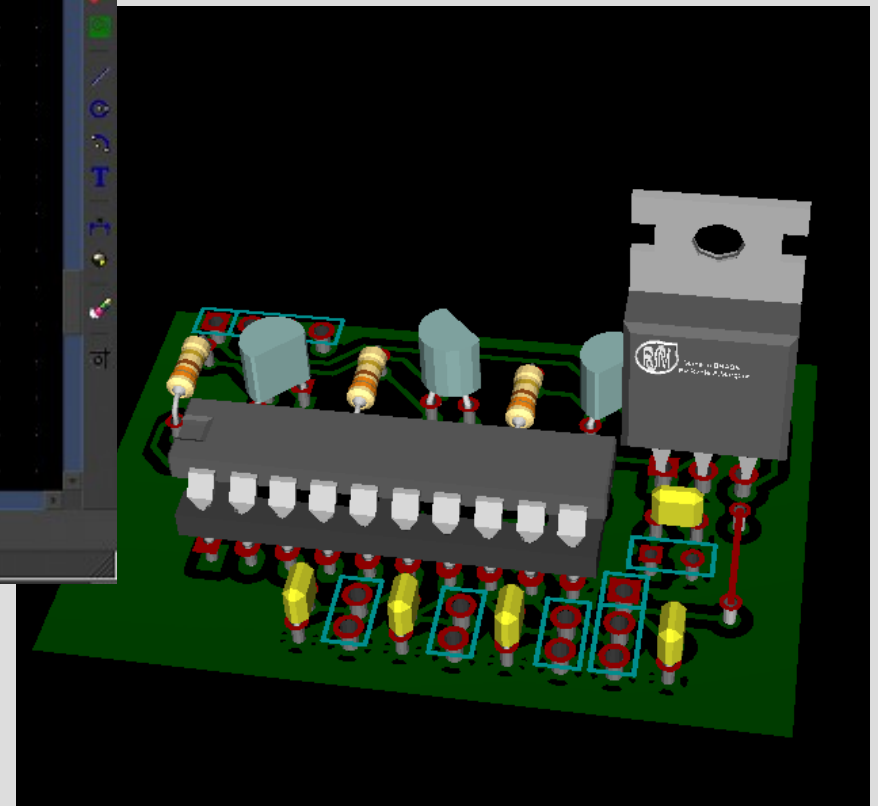
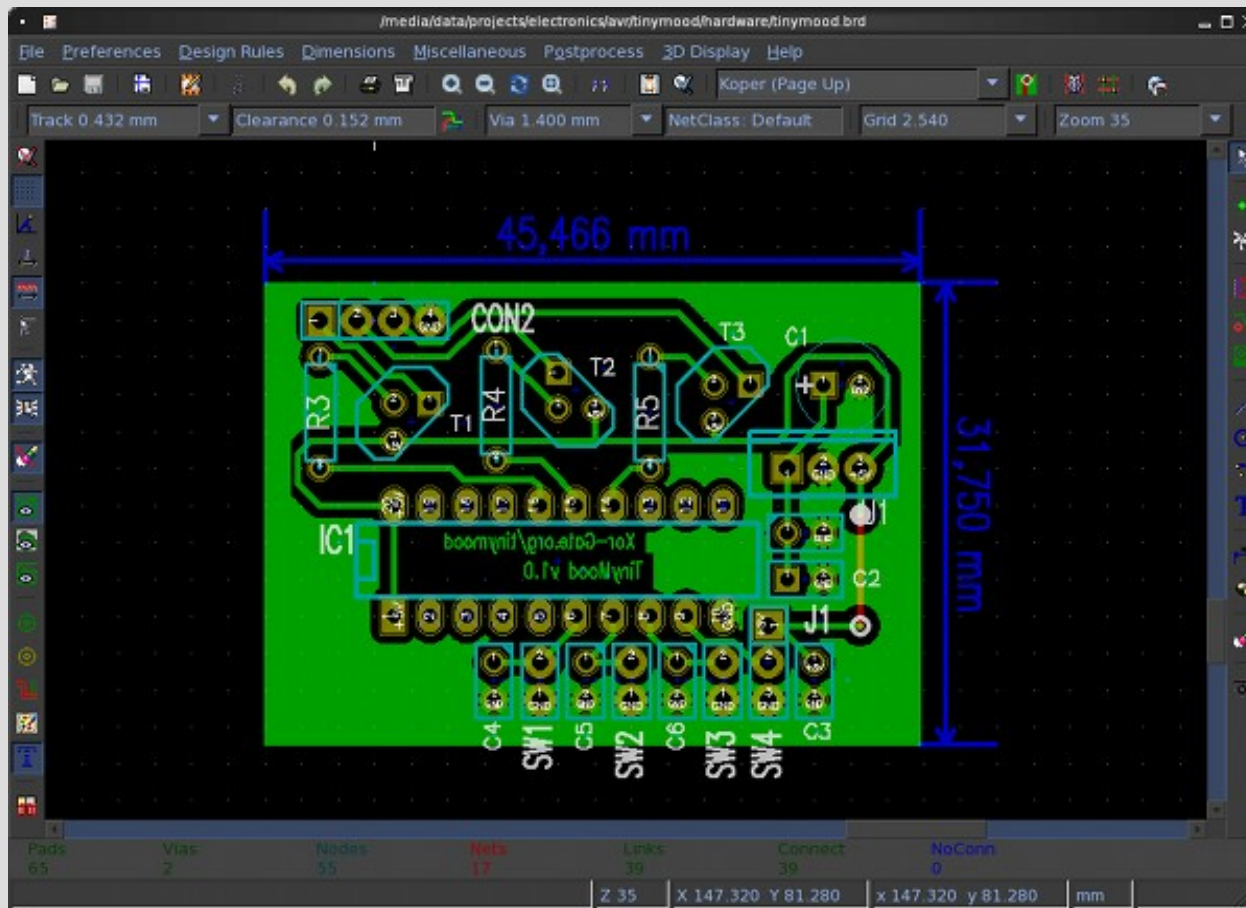
The screenshot displays the CvPCB software interface. The main window shows a component list with the following items:

1	C1	10uF/50V	: C1V7
2	C2	100nF	: C1
3	C3	100nF	: C1
4	C4	10nF	: C1
5	C5	10nF	: C1
6	C6	10nF	: C1
7	CON2	CONN_4	: SIL-4
8	IC1	ATTINY2313-P	: DIP-20_300_ELL
9	J1	CONN_2	: SIL-2
10	R3	330R	: R3
11	R4	330R	: R3
12	R5	330R	: R3
13	SW1	SW_PUSH	: SIL-2
14	SW2	SW_PUSH	: SIL-2
15	SW3	SW_PUSH	: SIL-2
16	SW4	SWITCH_INV	: SIL-3
17	T1	BC337	: T092
18	T2	BC337	: T092
19	T3	BC337	: T092
20	U1	7805	: T0220_VERT

The component list is highlighted in light blue. The component IC1 is selected, and its footprint assignment is shown in a separate window titled "Footprint: DIP-20_300_ELL". This window displays a 20-pin DIP package footprint with pins numbered 1 through 20. The footprint is labeled "DIP-20_300_ELL" and has a cyan border. The status bar at the bottom of the footprint window shows the following information:

U***	Last Change	Layer	Pads	Size	Orient
DIP-20_300_ELL	Jan 25, 1970	Component	20	0.0	0.0
Lib: /usr/loc	Z 22	X -0.6500	Y 0.3000	x -0.6500	y 0.3000

PCBNew – PCB Editor



PCBNew – PCB Editor

- Managing libraries of modules (footprints)
- Automatic and immediately modifications
- Ratsnest display
- Design Rule Check (D.R.C.)
- Polygon fill, with(out) thermal reliefs.
- Special UHF module creation
- 3D circuit viewer

PCBNew – PCB Editor

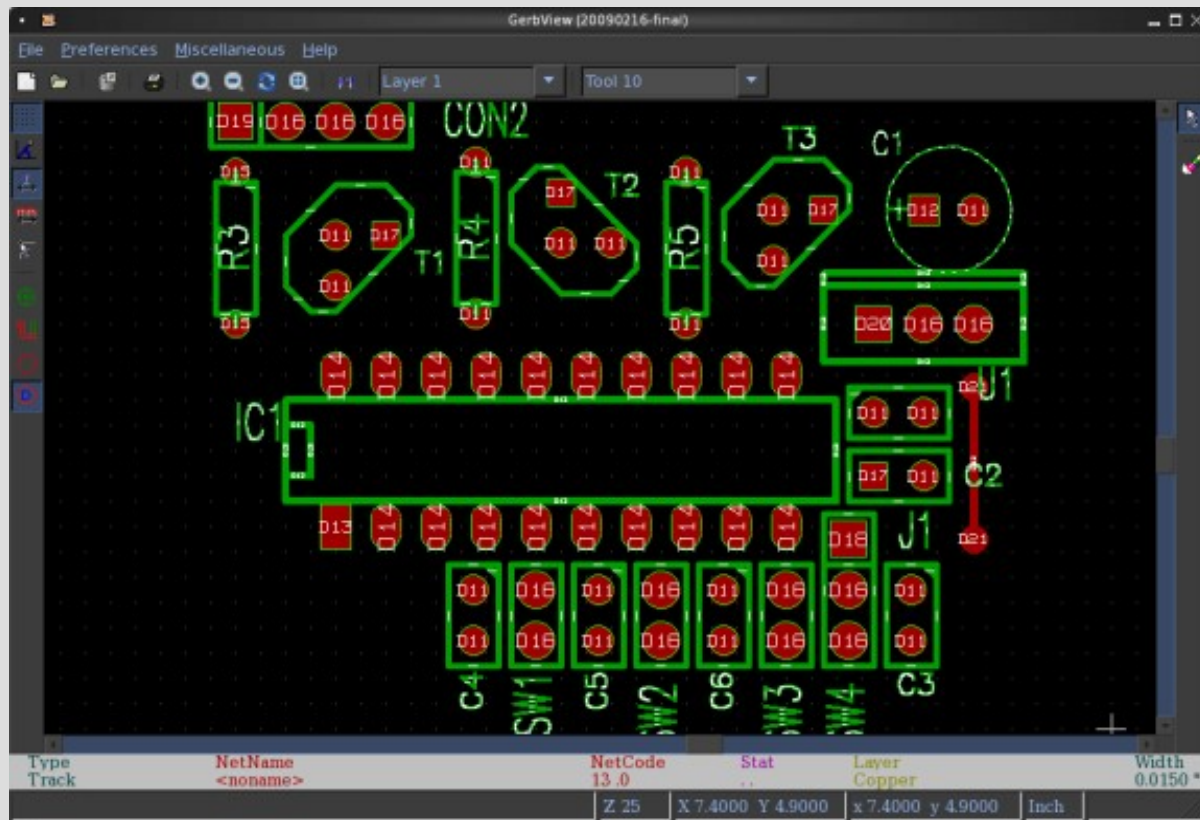
Output files for:

- Photoplotters in GERBER format
- Drilling in EXCELLON format and plans of drilling
- Tracing and drilling in HPGL format
- Tracing and drilling in POSTSCRIPT format

- And local printout.

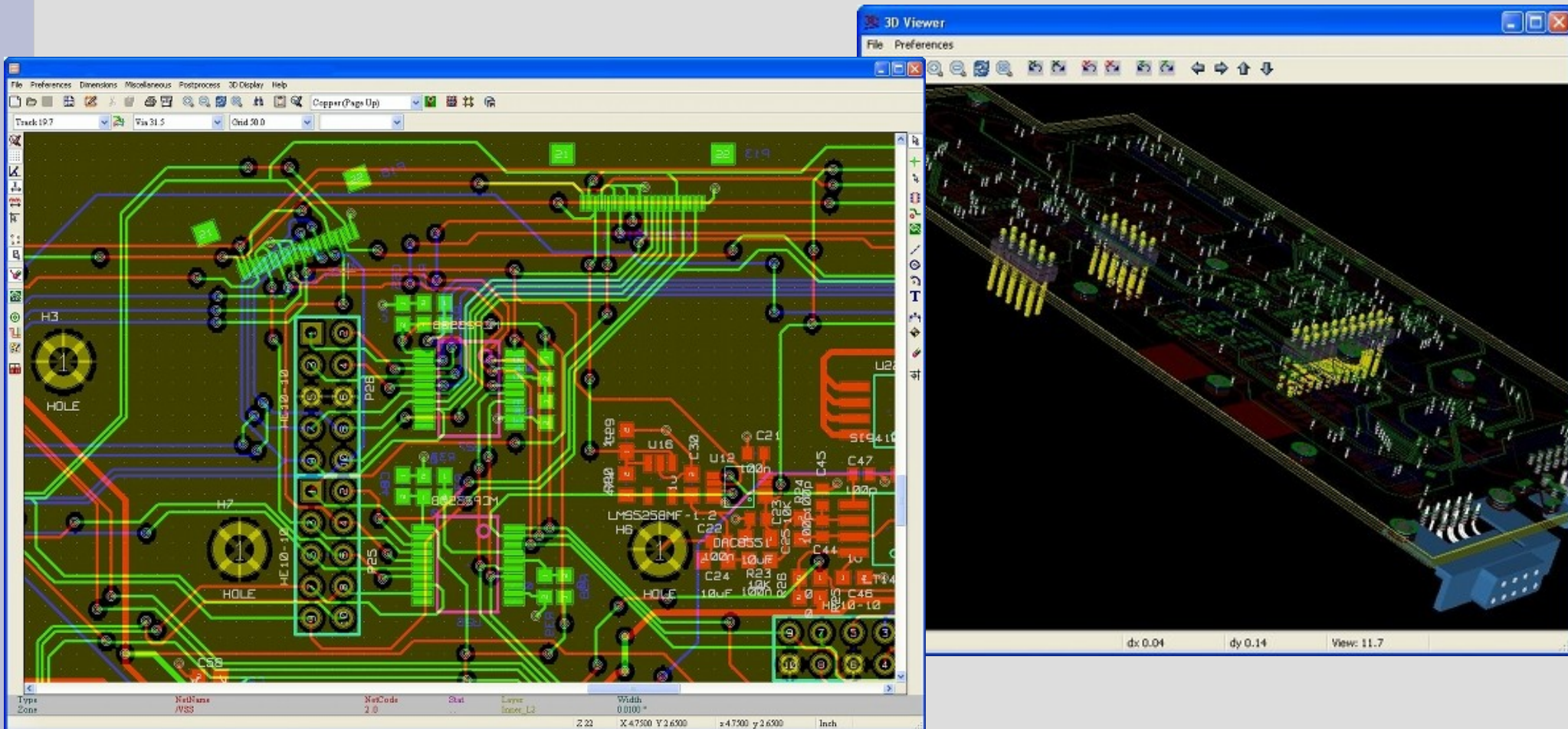
GerbView – Gerber Viewer

- Viewing rs274x gerber layers



Professional Use

admesy.nl - Advanced Measurement Systems
uses KiCad professional for 6 layer designs



Short Demo

:-)

Questions?

