Cvpcb.

Rubriques

1 -   Purpose

CVPCB allows you to assign each *component* that appears in the netlist created by the Schematic program to the name of the *footprint* which will represent it on a printed circuit board, and adds this information to the netlist.

Generally the netlist does not specify which *footprint* (i.e. the physical appearance of the component) the printed circuit software (**PCBNEW**) will have to display to create the general drawing of the card.

Components can be assigned to their corresponding footprints manually. You can create *Equivalence files*, which are look-up tables associating each component with its footprint. When equivalence files are available, automatic assignment is possible.

The list of the footprints available for the PCB software is contained in one or more **FOOTPRINT** *libraries.*

This interactive approach is much simpler than directly placing the assignments on the schematic, because as well as allowing automatic assignment, **CVPCB**, allows you to see the list of the footprints available, and to display them on the screen.

2 -   2 - General characteristics

2.1 - 2.1 - Characteristics

Cvpcb.
Interactive assignment of components with footprints or automatic assignment via equivalence files.
Generation (if necessary) of back- annotation files of this assignment to the schematic.

2.2 - 2.2 - Input netlist format

- **EESchema** (without footprint references).
- **VIEW LOGIC** WIRELIST.
- **VIEW LOGIC** NETLIST * (.NET+.PKG)

2.3 - 2.3 - Output Netlist

Two files are generated for *Pcbnew*:
- The enhanced **Netlist** file (with footprint references)
- An auxiliary component assignment file . (**.CMP**).

3 - 3 - Invoking Cvpcb

cvpcb (the file is then selected in CVPCB, via the file menu) or **cvpcb <filename>,** (*filename* being the name of the netlist file to be processed, from the schematic tool **Eeschema).**
The name of the file can be given with or without extension.
The extensions will if necessary be supplied by the defined in **cvpcb** configuration.
The two generated files will have the same name (with different extensions).
The standard extension of the file to be processed is **.net.**
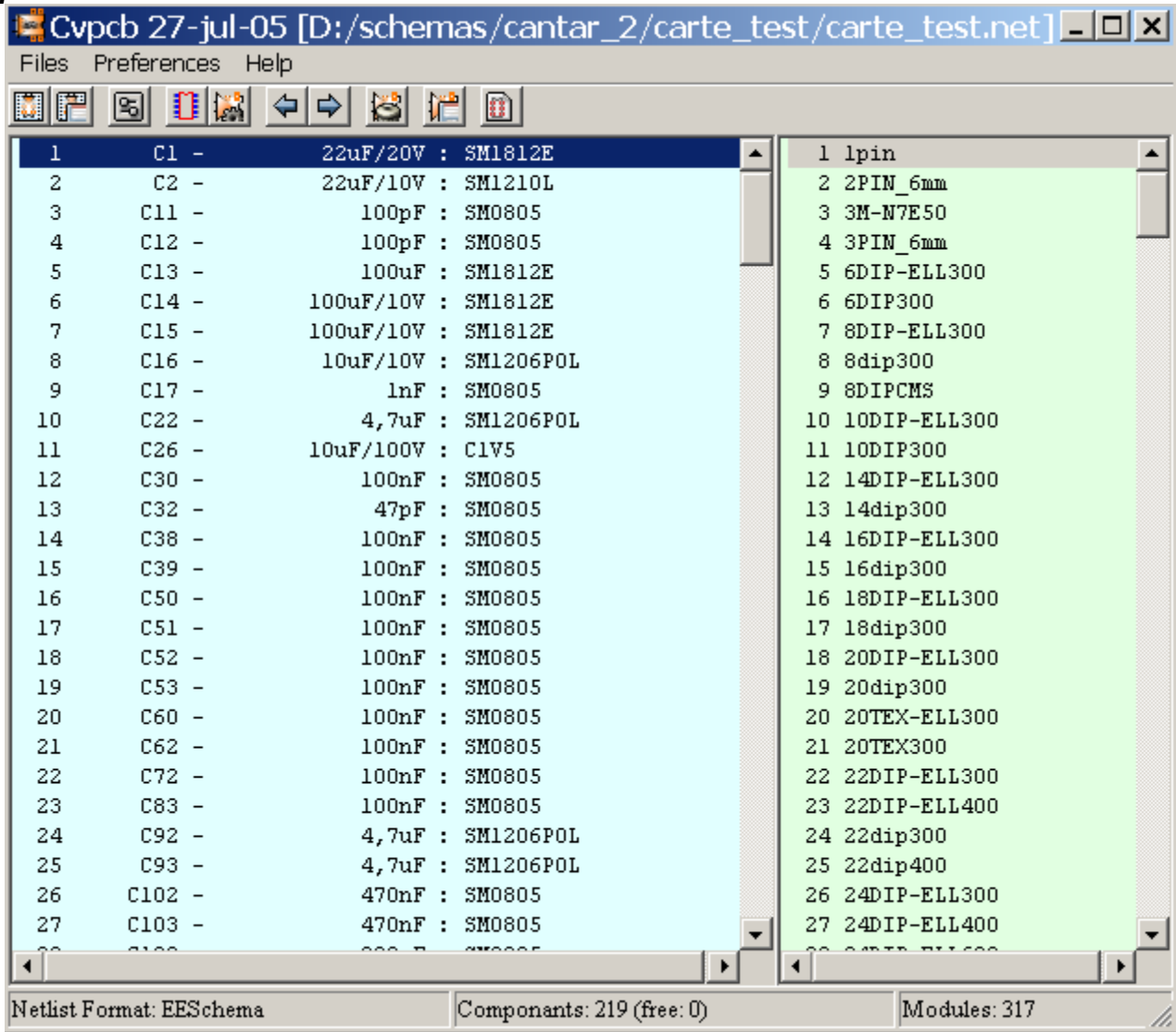The standard extension of the generated netlist file is **.net,** and will replace the old .net.
The standard extension of the file assigning components to the corresponding footprints (also generated by **CVPCB**) is **.cmp**.
These standard extensions can be modified by changing **Cvpcb** configuration.

4 - CVPCB Commands

4.1 - Main screen

Cvpcb.

The **Component** window (on the left) displays the list of components appearing in the Netlist read.

The *footprint* window (on the right) displays the list of footprints contained in the libraries read.

The **component** window is empty if no file is loaded, and the *footprint* window can be also empty if no footprint libraries are found.

4.2 - Main Screen Toolbar

The various functions are:

	Select the Netlist file to be processed.
	Create the assignment (stuff) file .CMP and the .NET file, the modified, expanded Netlist .
	Invoke the CVPCB configuration menu.
	Display the current footprint (i.e. whose name is highlighted in the footprint window).
	Automatically assign components/footprints starting from the equivalence files. Using this order obviously assumes that these files are available.
	Automatically run through the components towards the beginning of the list

|	
	until the first component not yet assigned a footprint.
⇨	Automatically run through the components towards the end of the list until the first component not yet assigned a footprint.
	Delete all assignments.
	Generate footprint assignment back-annotation file.
	Go to footprint documentation.

4.3 - CVPCB Configuration

4.3.1 - General screen

Invoking the configuration menu displays the following screen:
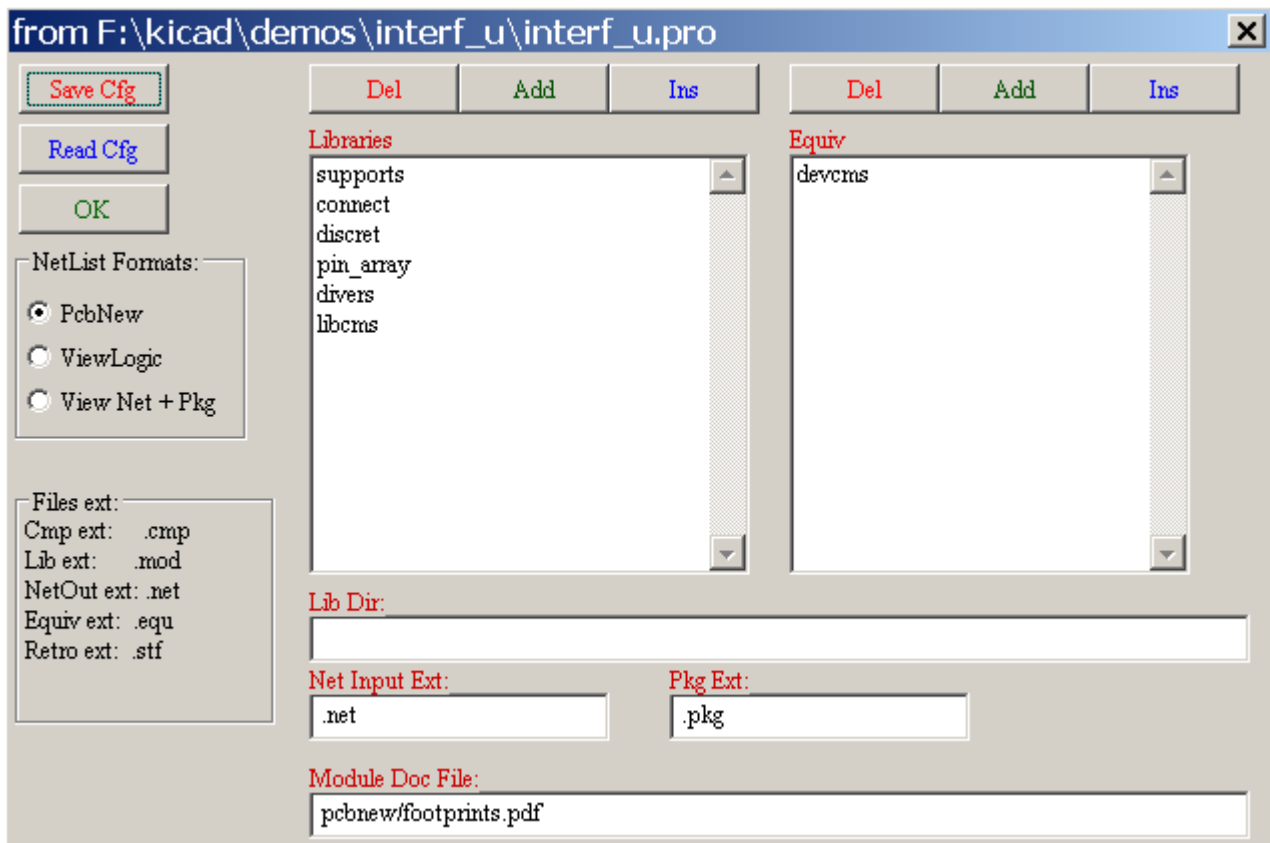
4.3.2 - Netlist Format

The options are:
PcbNew:
Read the netlist file generated by Eeschema, enhance it and generate the .cmp file.
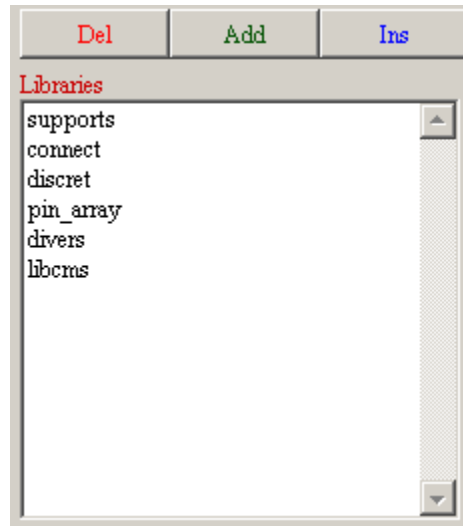**ViewLogic Wirelist:**
Accept standard format Viewlogic "Wirelist" (extension .wir) and generate a netlist file in PcbNew format (and the file .cmp)
ViewLogicNet & Pkg:
Accept the standard Viewlogic format netlist (.net) associated with the component file (.pkg) and generate a netlist file in Pcbnew format (and the .cmp file).

Cvpcb.

4.3.3 - Footprint <mark>Library</mark> selection

Del	Add	Ins

Libraries

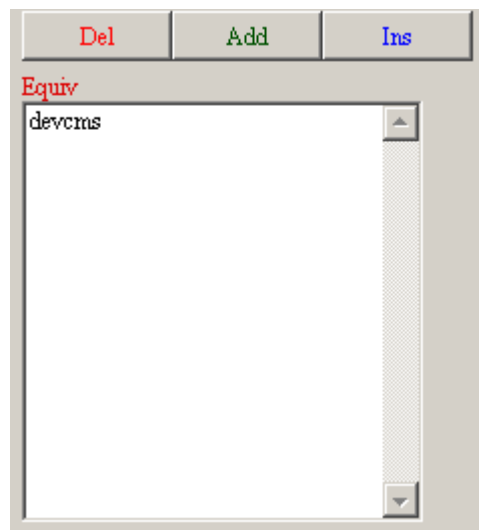supports
connect
discret
pin_array
divers
libcms

To select a file with the mouse:
- **Del** removes this name from the list.
- <mark>Add</mark> adds a new name to the list, **after the** selected name
- <mark>Ins</mark> adds a new name to the list, **before the** selected name.

Note:
Any modification of this list also affects <mark>*pcbnew*</mark>.

4.3.4 - Selecting the equivalence files

Del	Add	Ins

Equiv

devcms

To select with the mouse a file name.
- **Del** removes this name of the list.
- <mark>Add</mark> adds a new name to the list, **after the** selected name
- <mark>Ins</mark> adds a new name to the list, **before the** selected name

4.4 - Selecting library path and default extensions

Lib Dir:
f:\kicad\modules\

Net Input Ext: **Pkg Ext:**
.net   .pkg

<mark>Lib Dir</mark> is the default path of the footprint file (<mark>.mod</mark>) and of the equivalence (<mark>.equ</mark>) files.
If this field is left empty, the default path used will be <mark>*kicad/modules.*</mark>

Cvpcb.

4.5 - Viewing the current footprint

The <mark>Visu</mark> command allows display of the current footprint, i.e. the one that appears highlighted on the central line of the *footprint* window.
The various footprints can be displayed by clicking on the desired footprint (in the list of the footprints), as long as this window is in displayed.
One can also display the 3D view (if it has been created and assigned to the footprint)
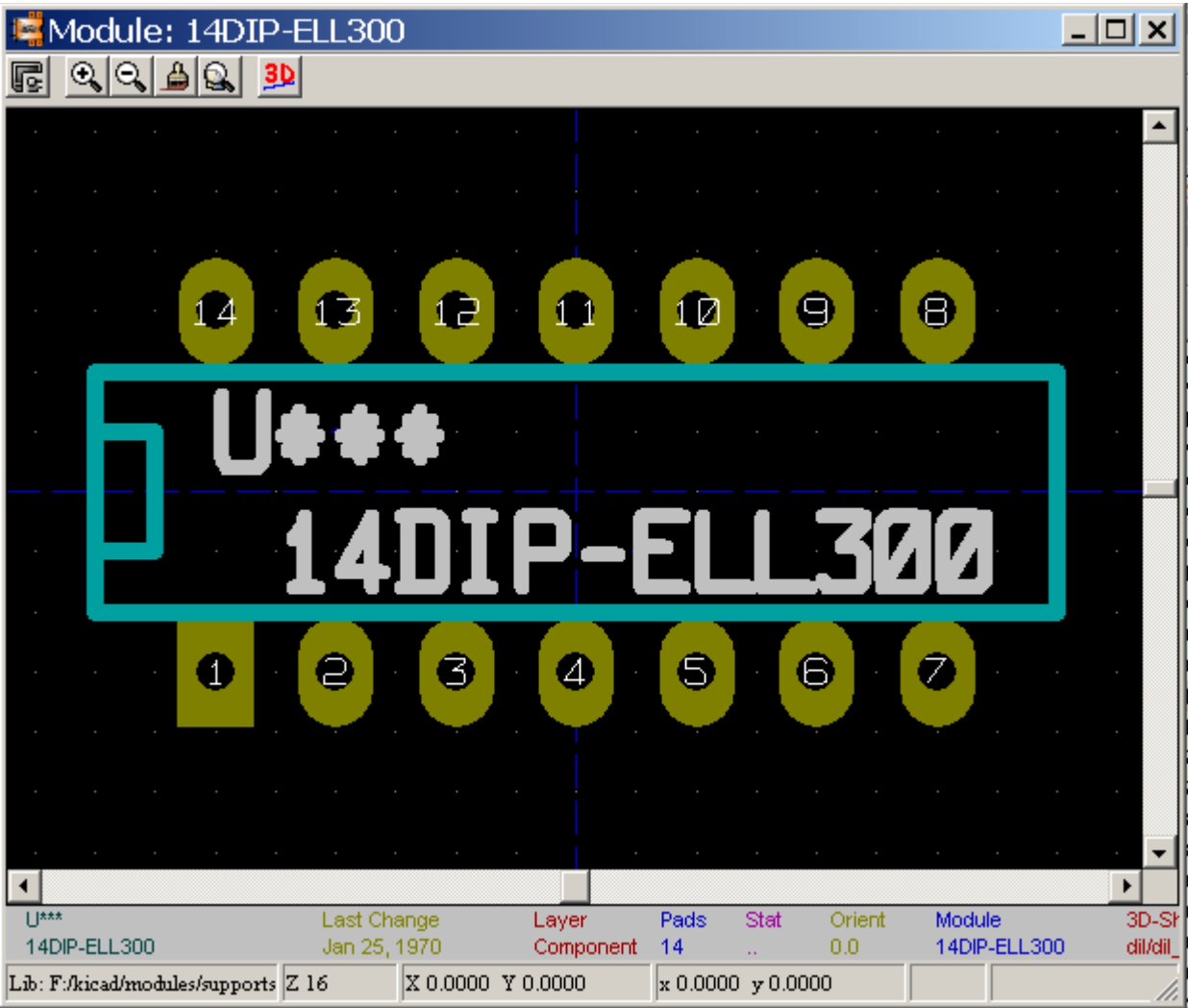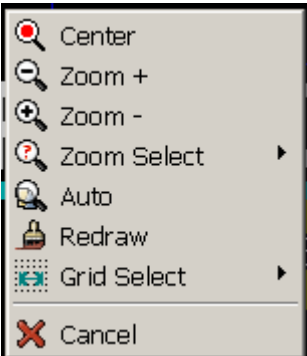

4.5.1 - Display

The co-ordinates of the cursor are displayed at the bottom of the screen :
Absolute co-ordinates (X <mark>nnnn</mark> Y <mark>nnnn</mark>) and relative co-ordinates (<mark>dx nnnn dy nnnn</mark>)
The relative co-ordinates are zeroed by the space bar.

4.5.2 - Keyboard Commands

F1	Zoom In
F2	Zoom Out
F3	Refresh Display
<space bar>:	Zero relative co-ordinates.

Cvpcb.

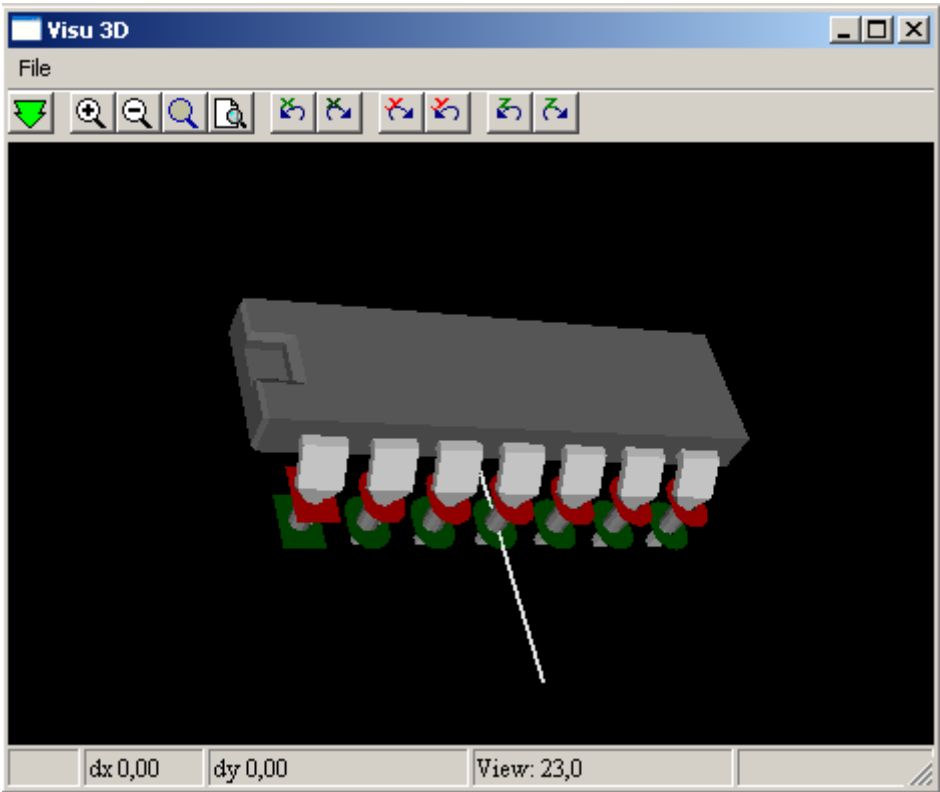## 4.5.3 - Pop Up Menu

Displayed by right-clicking the mouse:

Zoom Selection (Select Zoom)	Direct selection of the display zoom .
Grid Selection (Grid Select)	Direct selection of the grid.

4.5.4 - Toolbar

	Display Options
	Zoom Level
	3D Display

4.5.4 - 3D Display

5 -   Associating components with footprints

Cvpcb.

5.1 - Principle

In the footprint window double-click on the name of the desired **footprint** (This name is **highlighted)**, to assign it the **component** whose name is **highlighted on the central line** of the component window.
The next component in the list is selected:
- Automatically after an assignment.
- Manually using the mouse or cursor keys.

5.2 - Assignment

Double- click the **left mouse button** on the desired **footprint**

5.3 - Changing an existing assignment

This is done like a new assignment:
Double-click the **left mouse button** on the new desired **footprint**

6 -   Equivalence files

6.1 - Purpose

These files allow automatic assignment.
They give the name of the corresponding footprint according to the name (value) of the component.

6.2 - Format

They consist of a line for each component.
Each line has the following structure:
'component name' 'footprint name'
Each name being framed by the letter **',** the 2 names being separated by one or more spaces.
Example:
If the U3 component is circuit 14011 and its footprint is 14DIP300, the line is:
"14011" "14DIP300"

7 -   Back- annotation file

This a file can be used for back- annotation of a schematic but is not used by PCBNEW.
It consists of a line for each component, giving the name of the footprint according to its reference.
Example:
If the U3 component was assigned the footprint14DIP300, the generated line is
comp "U3" = footprint "14DIP300"
The file created has the root name of the CVPCB input file, with extension **.stf**, and is placed in the same folder as the generated netlist.