Cvpcb.

Headings

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 -   General characteristics

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 - Input netlist format

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

2.3 - Output Netlist

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

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.**

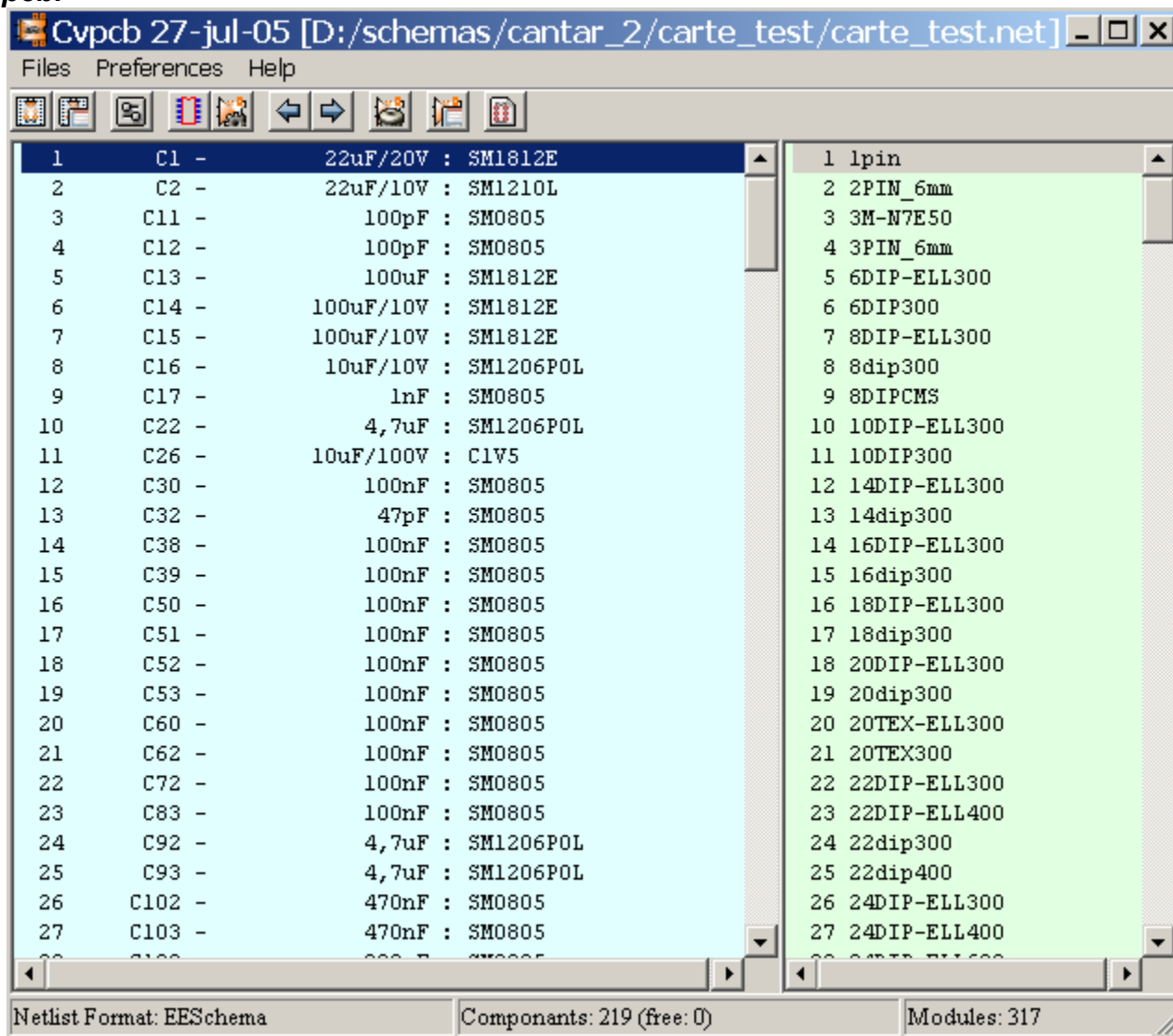The standard extension of the generated netlist file is **.net,** and will replace the old .net.

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.

```
Cvpcb 27-jul-05 [D:/schemas/cantar_2/carte_test/carte_test.net]   _ □ X
Files   Preferences   Help

1      C1  -       22uF/20V : SM1812E          ▲   1  1pin                    ▲
2      C2  -       22uF/10V : SM1210L              2  2PIN_6mm
3      C11 -          100pF : SM0805               3  3M-N7E50
4      C12 -          100pF : SM0805               4  3PIN_6mm
5      C13 -          100uF : SM1812E              5  6DIP-ELL300
6      C14 -      100uF/10V : SM1812E              6  6DIP300
7      C15 -      100uF/10V : SM1812E              7  8DIP-ELL300
8      C16 -       10uF/10V : SM1206POL            8  8dip300
9      C17 -            1nF : SM0805               9  8DIPCMS
10     C22 -          4,7uF : SM1206POL           10  10DIP-ELL300
11     C26 -      10uF/100V : C1V5                11  10DIP300
12     C30 -          100nF : SM0805              12  14DIP-ELL300
13     C32 -           47pF : SM0805              13  14dip300
14     C38 -          100nF : SM0805              14  16DIP-ELL300
15     C39 -          100nF : SM0805              15  16dip300
16     C50 -          100nF : SM0805              16  18DIP-ELL300
17     C51 -          100nF : SM0805              17  18dip300
18     C52 -          100nF : SM0805              18  20DIP-ELL300
19     C53 -          100nF : SM0805              19  20dip300
20     C60 -          100nF : SM0805              20  20TEX-ELL300
21     C62 -          100nF : SM0805              21  20TEX300
22     C72 -          100nF : SM0805              22  22DIP-ELL300
23     C83 -          100nF : SM0805              23  22DIP-ELL400
24     C92 -          4,7uF : SM1206POL           24  22dip300
25     C93 -          4,7uF : SM1206POL           25  22dip400
26     C102 -         470nF : SM0805              26  24DIP-ELL300
27     C103 -         470nF : SM0805          ▼   27  24DIP-ELL400       ▼
◄                                          ►    ◄                        ►
Netlist Format: EESchema      Components: 219 (free: 0)      Modules: 317
```
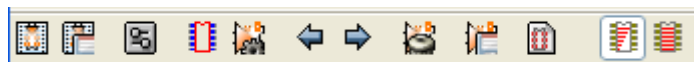
The *Component* window (on the left) displays the list of components appearing in the Netlist read.

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.

Cvpcb.

⬅	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.
📑 📑	Allows or forbids the footprint filtering to display the list of footprints When the footprint filtering is allowed, the list of footprints shows only the autorized footprints for the current selected component.

4.3 - CVPCB Configuration

4.3.1 - General screen

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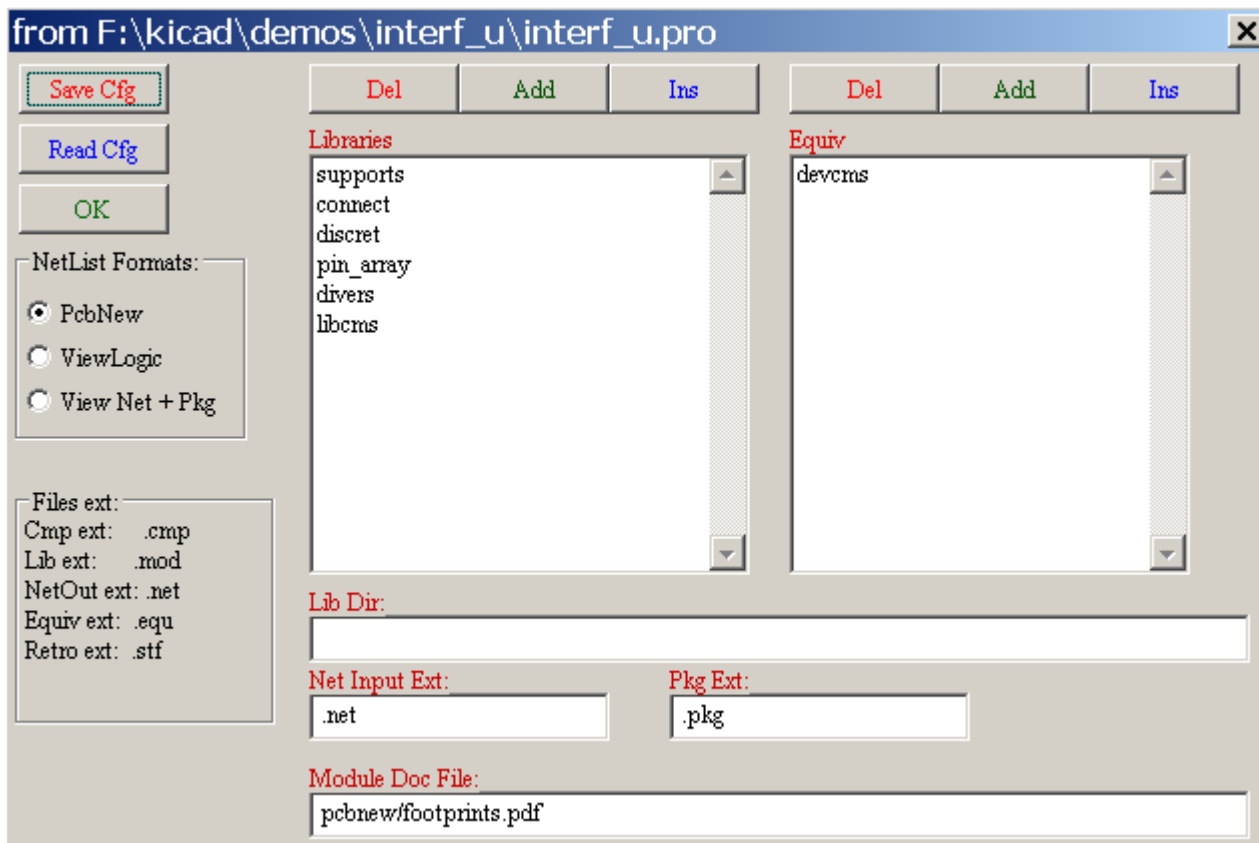
4.3.2 - Netlist Format

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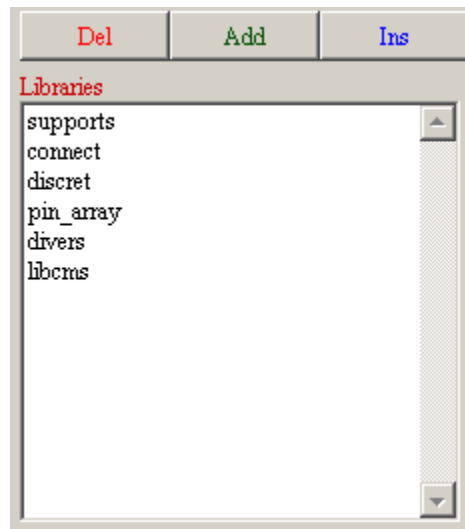*ViewLogic Wirelist:*
Accept standard format Viewlogic "Wirelist" (extension .wir) and generate a netlist file in PcbNew format (and the file .cmp)
ViewLogicNet & Pkg:
Accept the standard Viewlogic format netlist (.net) associated with the component file (.pkg) and generate a netlist file in Pcbnew format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

Cvpcb.

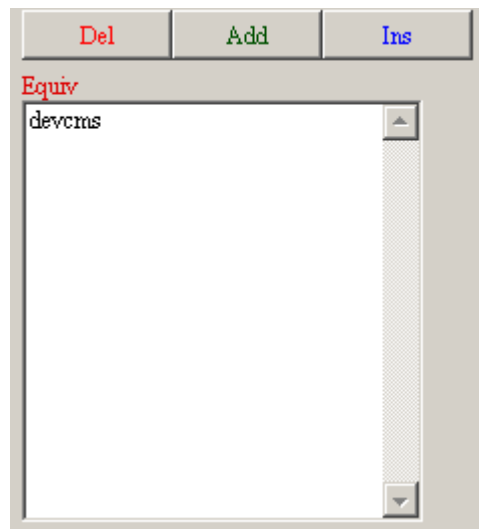### 4.3.3 - Footprint Library selection

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

Note:
Any modification of this list also affects ***pcbnew***.

4.3.4 - Selecting the equivalence files

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

4.4 - Selecting library path and default extensions

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

4.5 - Viewing the current footprint

The Visu 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)
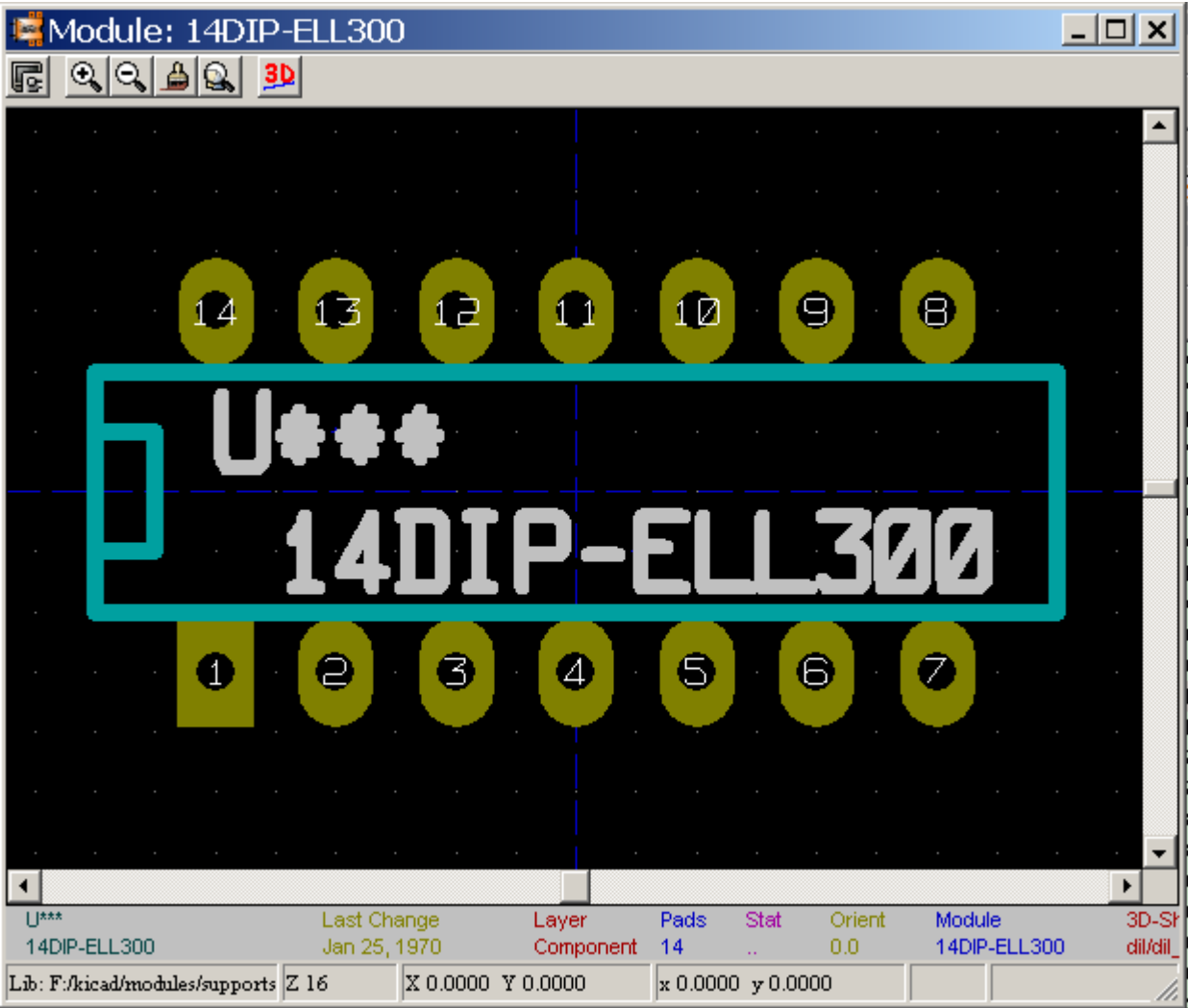
4.5.1 - Display

The co-ordinates of the cursor are displayed at the bottom of the screen :

Absolute co-ordinates (X nnnn Y nnnn) and relative co-ordinates (dx nnnn dy nnnn)

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
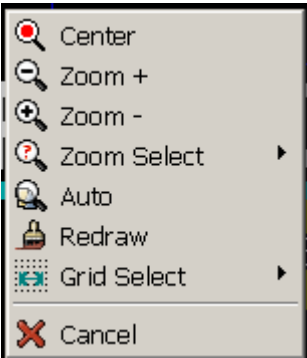
Displayed by right-clicking the mouse:

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

5.1 - Principle

Cvpcb.

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.

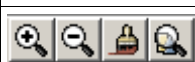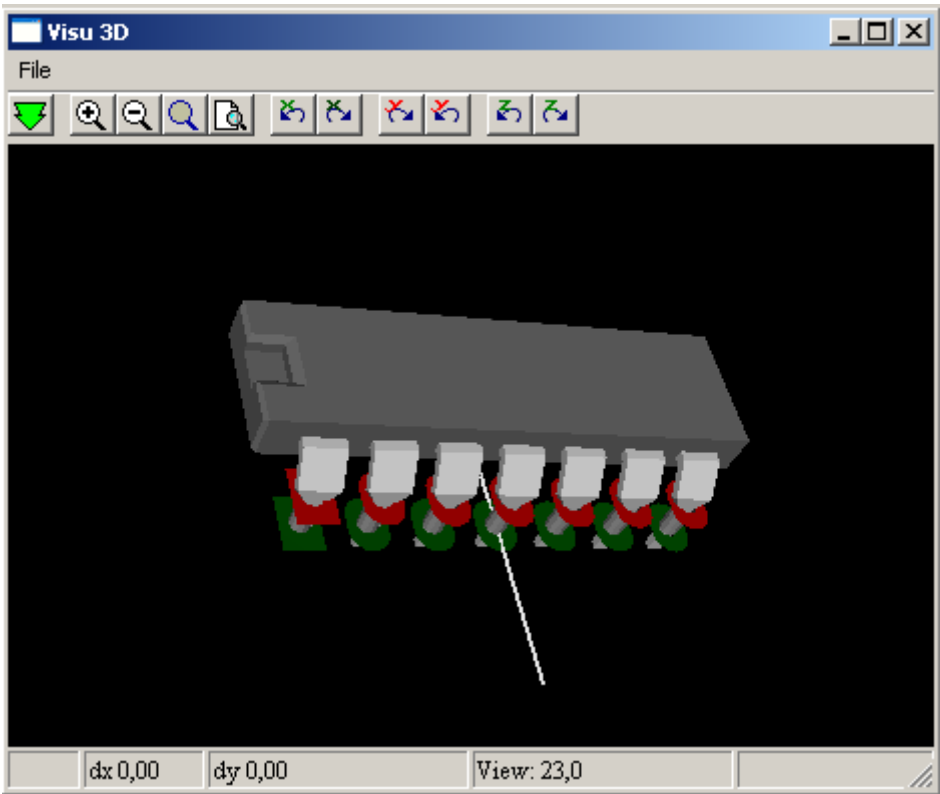The next component in the list is selected:
- Automatically after an assignment.
- Manually using the mouse or cursor keys.

5.2 - Assignment

Double- click the **left mouse button** on the desired **footprint**
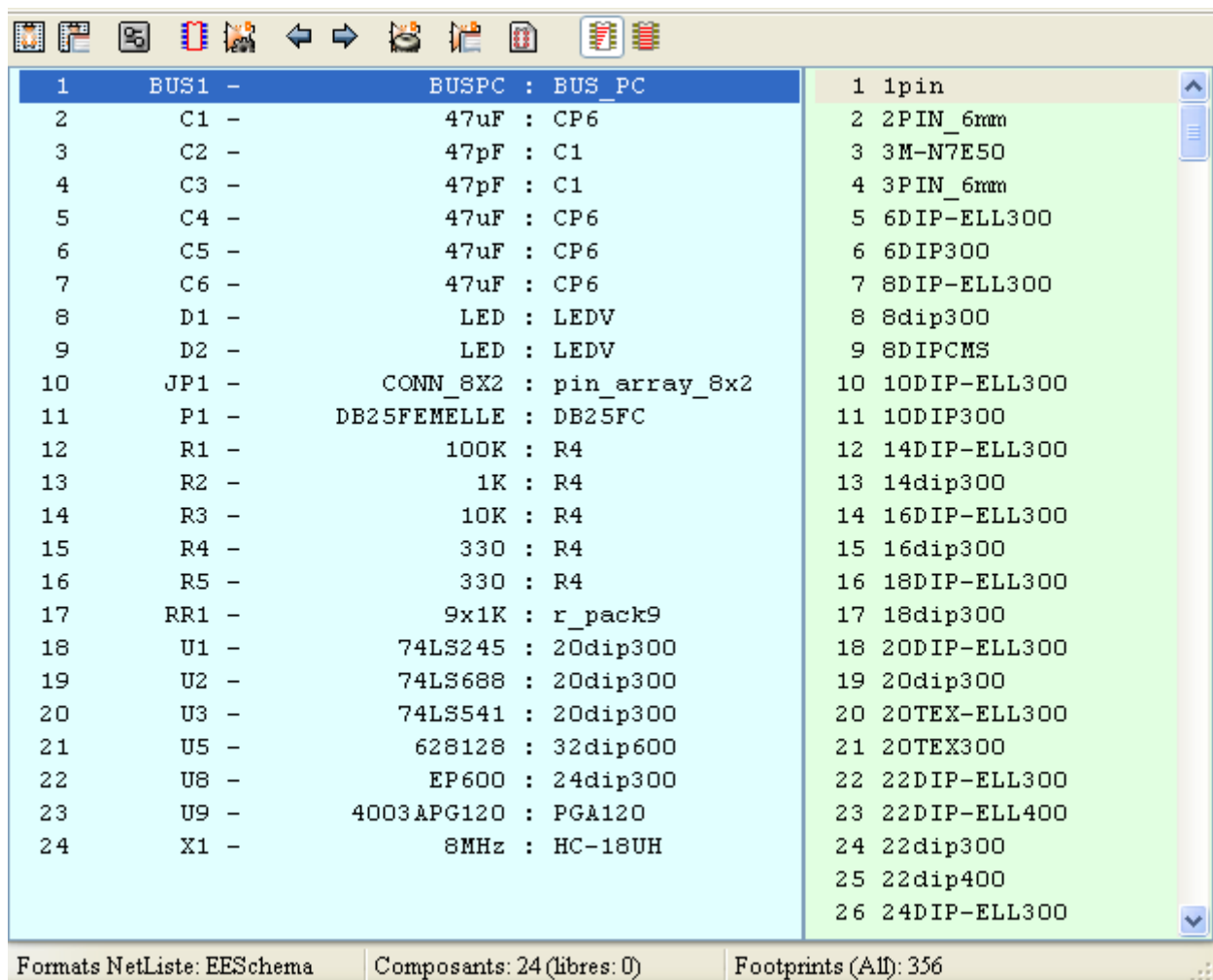
5.3 - Changing an existing assignment

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

5.4 - Filtering the footprint list:

If the selected component has an allowed footprint list, the displayed footprint list in Cvpcb is filtered according to this list.
Without filtering:

```
 1      BUS1 -              BUSPC : BUS_PC          1  1pin
 2       C1 -                47uF : CP6             2  2PIN_6mm
 3       C2 -                47pF : C1              3  3M-N7E50
 4       C3 -                47pF : C1              4  3PIN_6mm
 5       C4 -                47uF : CP6             5  6DIP-ELL300
 6       C5 -                47uF : CP6             6  6DIP300
 7       C6 -                47uF : CP6             7  8DIP-ELL300
 8       D1 -                 LED : LEDV            8  8dip300
 9       D2 -                 LED : LEDV            9  8DIPCMS
10      JP1 -             CONN_8X2 : pin_array_8x2 10  10DIP-ELL300
11       P1 -         DB25FEMELLE : DB25FC         11  10DIP300
12       R1 -                100K : R4             12  14DIP-ELL300
13       R2 -                  1K : R4             13  14dip300
14       R3 -                 10K : R4             14  16DIP-ELL300
15       R4 -                 330 : R4             15  16dip300
16       R5 -                 330 : R4             16  18DIP-ELL300
17      RR1 -                9x1K : r_pack9        17  18dip300
18       U1 -              74LS245 : 20dip300      18  20DIP-ELL300
19       U2 -              74LS688 : 20dip300      19  20dip300
20       U3 -              74LS541 : 20dip300      20  20TEX-ELL300
21       U5 -               628128 : 32dip600      21  20TEX300
22       U8 -                EP600 : 24dip300      22  22DIP-ELL300
23       U9 -          4003APG120 : PGA120         23  22DIP-ELL400
24       X1 -                 8MHz : HC-18UH       24  22dip300
                                                   25  22dip400
                                                   26  24DIP-ELL300

Formats NetListe: EESchema    Composants: 24 (libres: 0)    Footprints (All): 356
```

Cvpcb.

With filtering:

1	BUS1 –	BUSPC	:	BUS_PC	1	R1
2	C1 –	47uF	:	CP6	2	R3
3	C2 –	47pF	:	C1	3	R4
4	C3 –	47pF	:	C1	4	R5
5	C4 –	47uF	:	CP6	5	R6
6	C5 –	47uF	:	CP6	6	R7
7	C6 –	47uF	:	CP6	7	SM0603
8	D1 –	LED	:	LEDV	8	SM0805
9	D2 –	LED	:	LEDV		
10	JP1 –	CONN_8X2	:	pin_array_8x2		
11	P1 –	DB25FEMELLE	:	DB25FC		
12	R1 –	100K	:	R4		
13	R2 –	1K	:	R4		
14	R3 –	10K	:	R4		
15	R4 –	330	:	R4		
16	R5 –	330	:	R4		
17	RR1 –	9x1K	:	r_pack9		
18	U1 –	74LS245	:	20dip300		
19	U2 –	74LS688	:	20dip300		
20	U3 –	74LS541	:	20dip300		
21	U5 –	628128	:	32dip600		
22	U8 –	EP600	:	24dip300		
23	U9 –	4003APG120	:	PGA120		
24	X1 –	8MHz	:	HC-18UH		

Formats NetListe: EESchema   Composants: 24 (libres: 0)   Footprints (filtered): 8

Under Eeschema, the allowed footprint list was:

Propriétés pour R

Options | Doc | Alias | Champs | Filtrage Modules

Modules

R?
SM0603
SM0805

Ajouter

Supprimer

Tout Supprimer

Annuler   OK
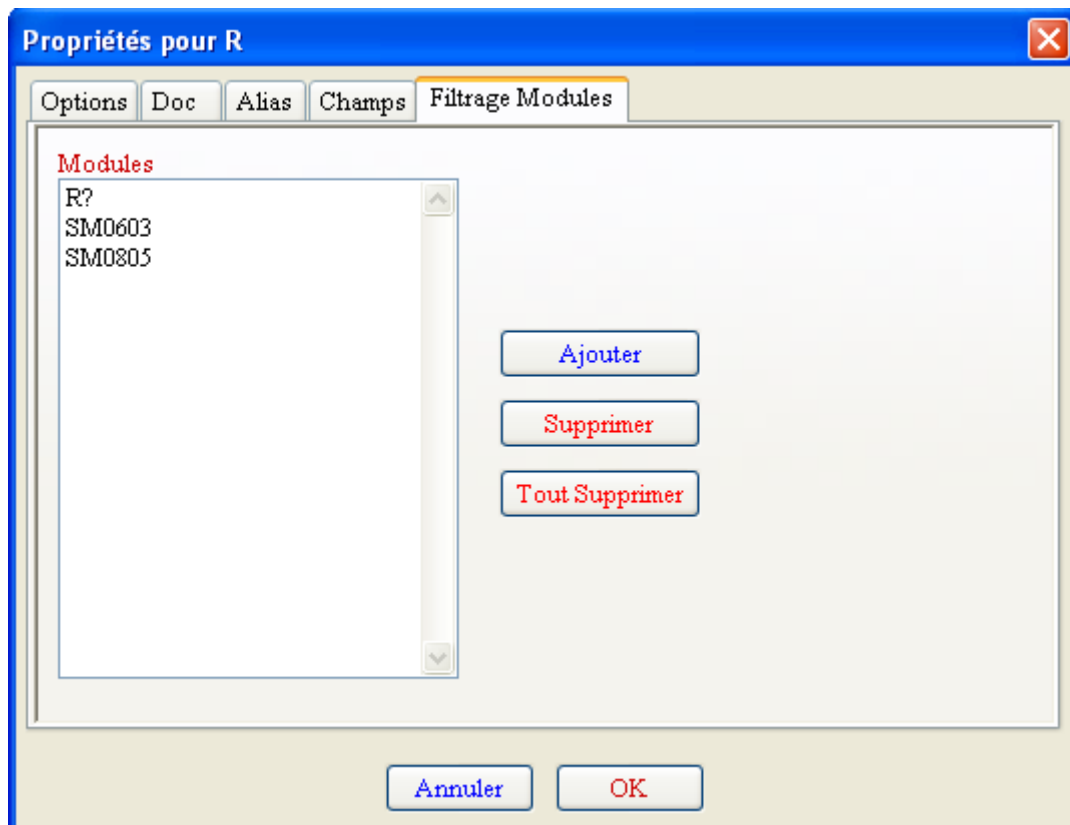
Cvpcb.

The icone allows or .forbids the filtering
When the filtering is not allowed, the full footprint list is shown.

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.

This is useful to import the footprints names of components used within a schematic in Eeschema, before creation a file report for instance.
This is not necessary to work with Pcbnew.