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.**

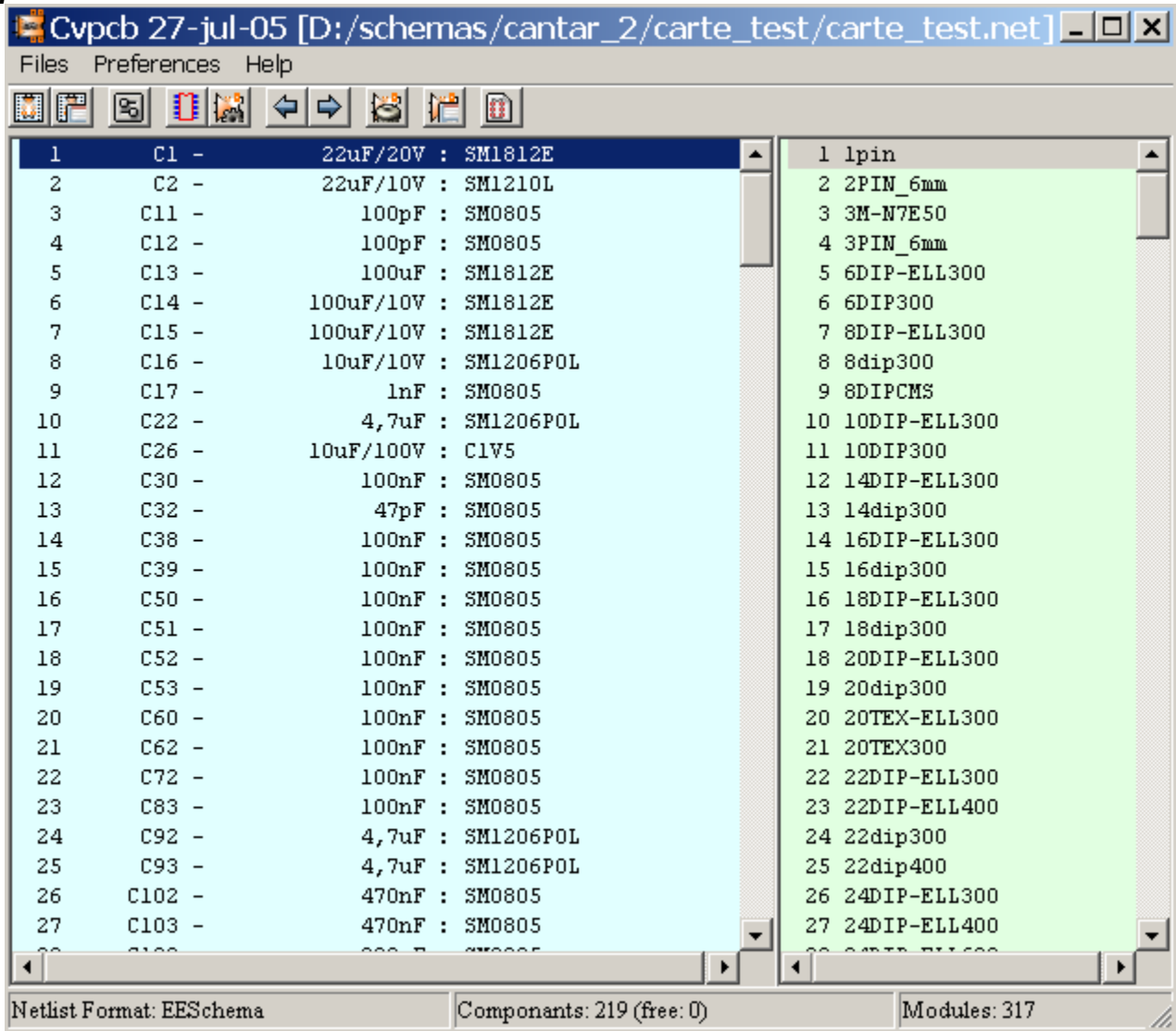The standard extension of the generated netlist file is **.net,** and will replace the old .net.

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.

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
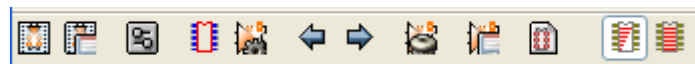
The various functions are:

	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.
🖼🖼	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:

Cvpcb.
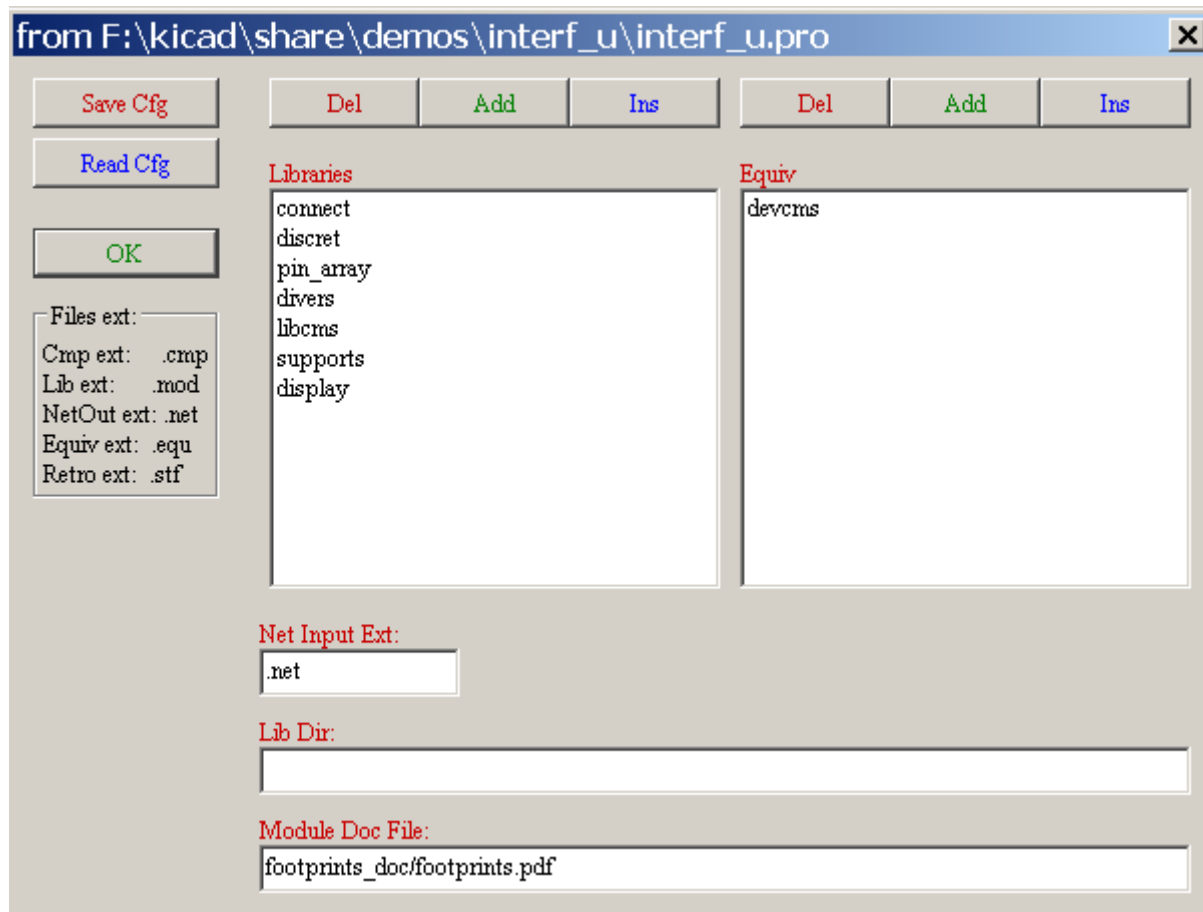
4.3.2 - Footprint Library selection

Del	Add	Ins

Libraries

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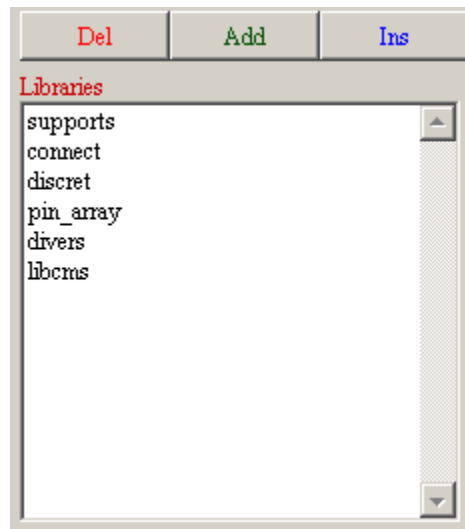
To select a file with the mouse:
- **Del** removes this name from the list.
- **Add** adds a new name to the list, **after the** selected name
- **Ins** adds a new name to the list, **before the** selected name.

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

4.3.3 - Selecting the equivalence files

Del	Add	Ins

Equiv

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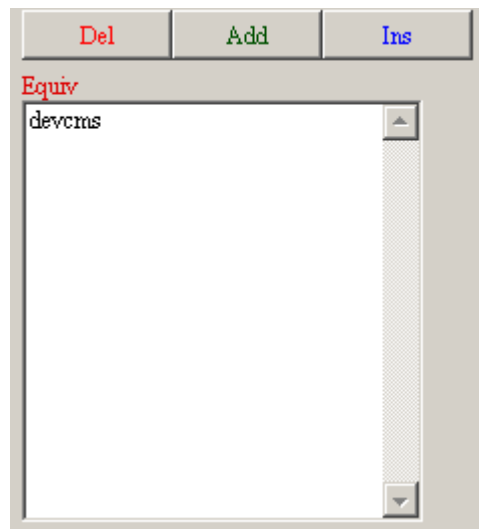
To select with the mouse a file name.
- **Del** removes this name of the list.
- **Add** adds a new name to the list, **after the** selected name
- **Ins** adds a new name to the list, **before the** selected name

4.4 - Selecting library path and default extensions

Cvpcb.

Net Input Ext:

.net

Lib Dir:

Module Doc File:

footprints_doc/footprints.pdf

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/share/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)
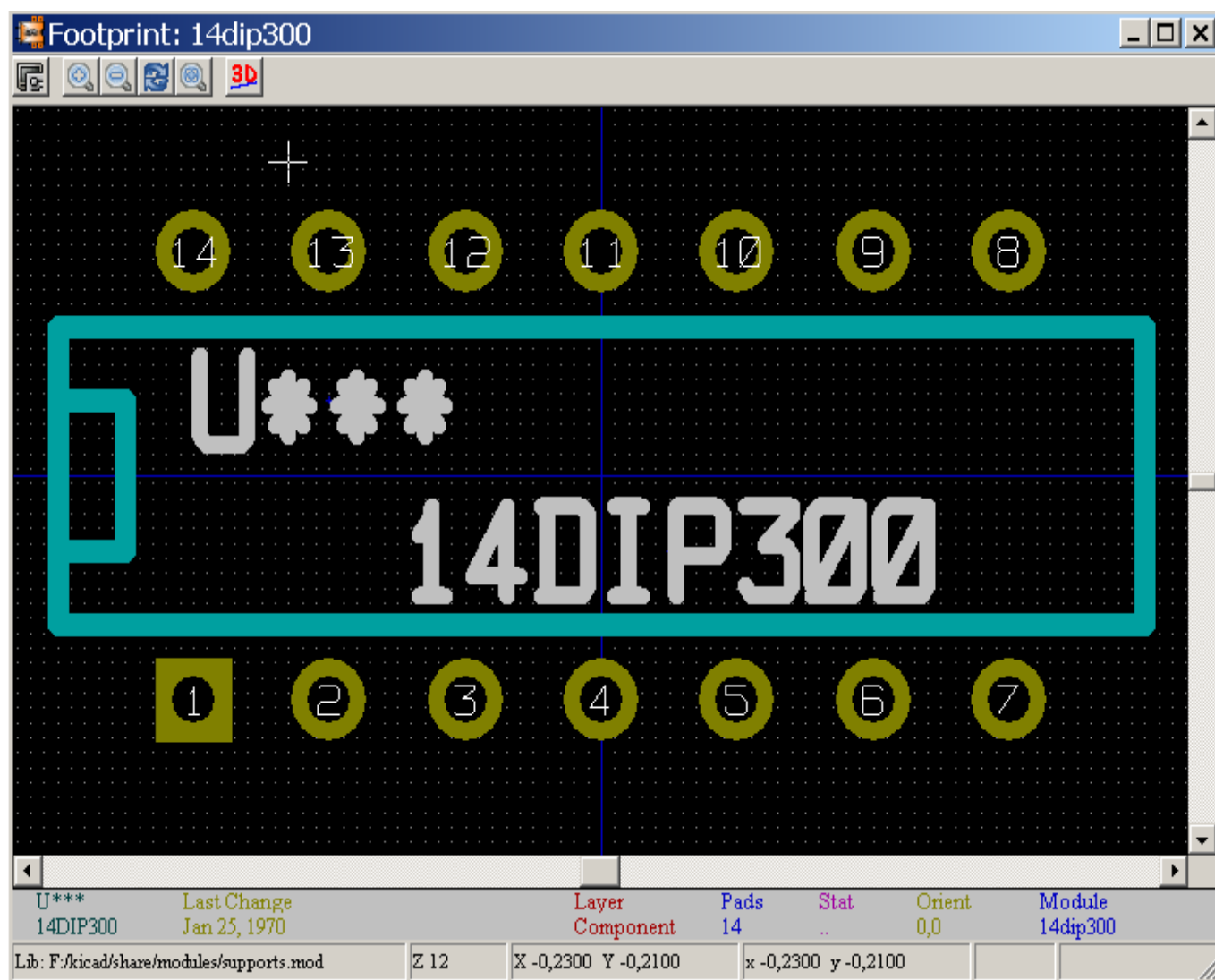
4.5.1 - Display

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.

Cvpcb.

4.5.2 - Keyboard Commands

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

4.5.3 - Pop Up Menu
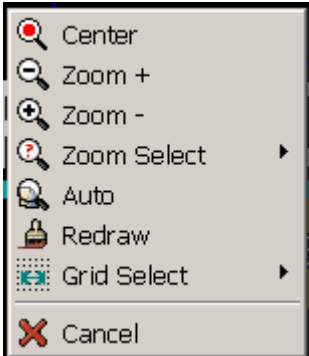
Displayed by right-clicking the mouse:

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

4.5.5 - 3D Display
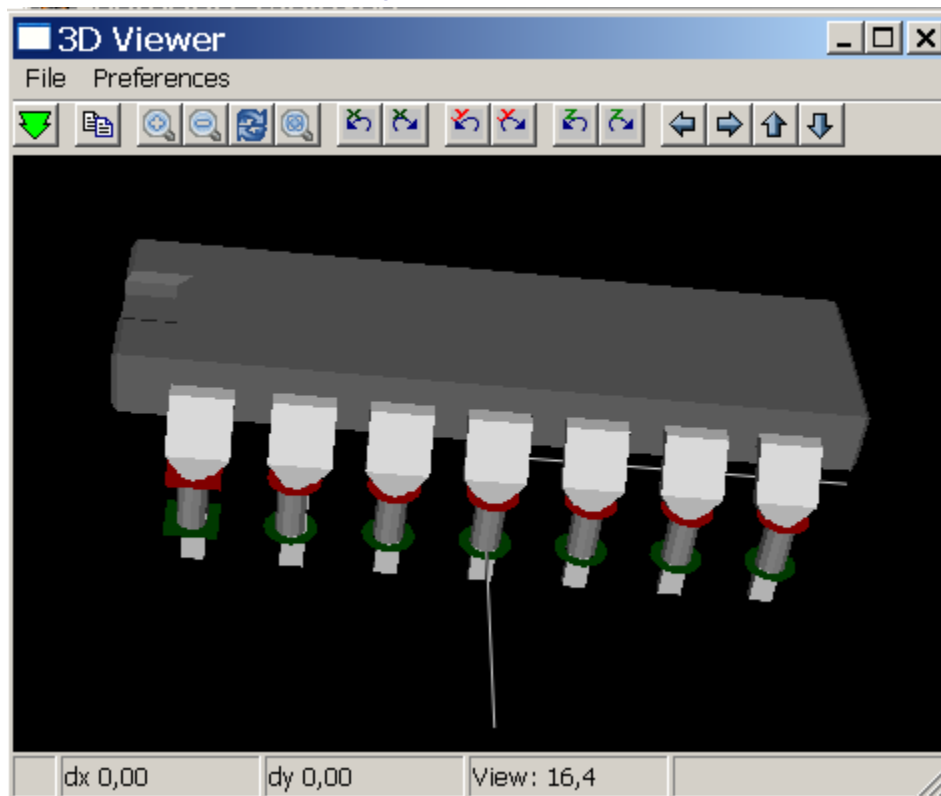
5 - Associating components with footprints

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**

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.

Cvpcb.

Without filtering:

#	Ref	Value	Footprint		#	Footprint
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
```
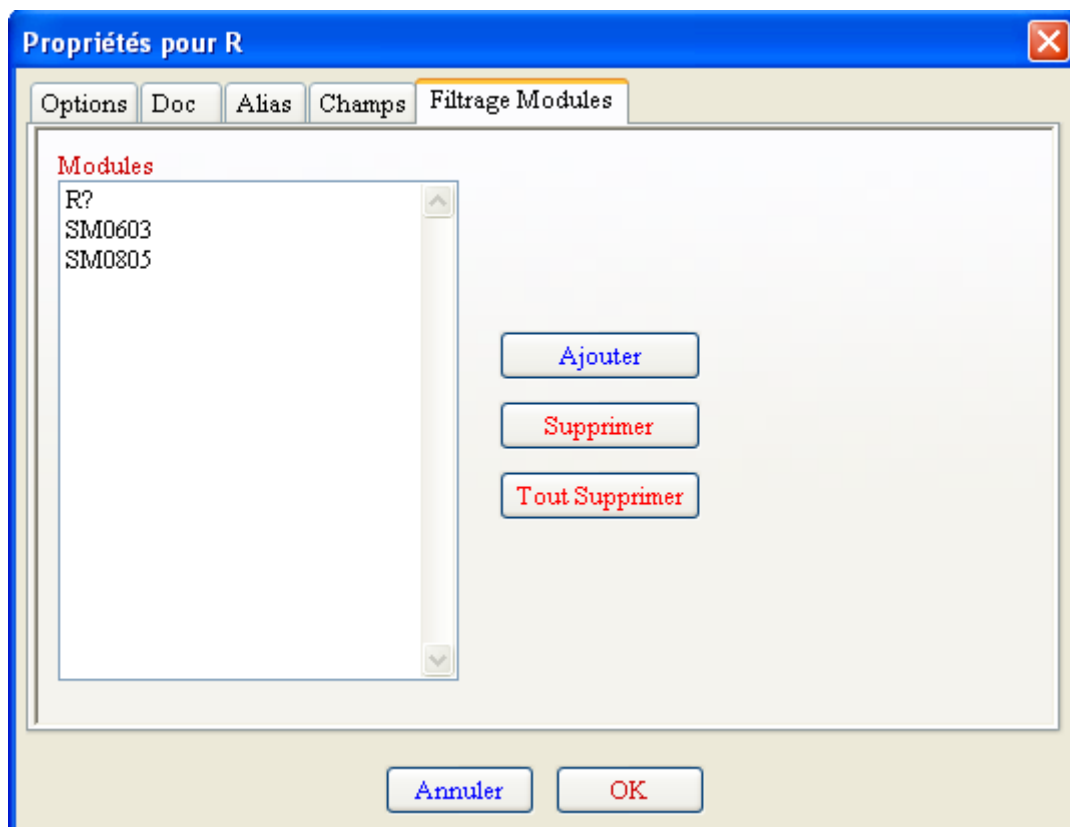
Formats NetListe: EESchema   Composants: 24 (libres: 0)   Footprints (filtered): 8

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 -   Automatic associations

6.1 - Equivalence files

These files allow automatic assignment.
They give the name of the corresponding footprint according to the name (*value*) of the component.
These files have the standard extension *.equ*
By selecting suitable files for a given, project, it is easy to use different technologies (like smd,or dip packages …)
See "*Selecting the equivalence files*".

6.2 - Format

They consist of a line for each component.
Each line has the following structure:
'component value' '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'
A line starting by **#** is a comment.
Here is a sample:

```
#integrated circuits (smd):
'74LV14' 'SO14E'
'74HCT541M' 'SO20L'
'EL7242C' 'SO8E'
'DS1302N' 'SO8E'
'XRC3064' 'VQFP44'
'LM324N' 'SO14E'
'LT3430' 'SSOP17'
'LM358' 'SO8E'
'LTC1878' 'MSOP8'
'24LC512I/SM' 'SO8E'
'LM2903M' 'SO8E'
'LT1129_SO8' 'SO8E'
'LT1129CS8-3.3' 'SO8E'
'LT1129CS8' 'SO8E'
'LM358M' 'SO8E'
'TL7702BID' 'SO8E'
'TL7702BCD' 'SO8E'
'U2270B' 'SO16E'
#Xilinx
'XC3S400PQ208' 'PQFP208'
'XCR3128-VQ100' 'VQFP100'
'XCF08P' 'BGA48'

#upro
'MCF5213-LQFP100' 'VQFP100'

#regulators
'LP2985LV'      'SOT23-5'
```

Cvpcb.

6.3 - Association:

The automatic association is launch by:

All components found (by their value) in a *.equ file will have their footprint automatically select.

7 - Back- annotation file

This 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.