

Headings

[1 - Purpose](#)

[2 - General characteristics](#)

[2.1 - Characteristics](#)

[2.2 - Input files](#)

[2.3 - Output files](#)

[3 - Invoking Cvpcb](#)

[4 - CVPCB Commands](#)

[4.1 - Main screen](#)

[4.2 - Main Screen Toolbar](#)

[4.3 - CVPCB Configuration](#)

[4.3.1 - General screen](#)

[4.3.2 - Footprint Library selection](#)

[4.3.3 - Selecting the equivalence files](#)

[4.4 - Selecting default library path.](#)

[4.4.1 - Search paths:](#)

[4.4.2 - Path added by users:](#)

[4.4.3 - Paths automatically set by Cvpcb](#)

[4.5 - Viewing the current footprint](#)

[4.5.1 - Display](#)

[4.5.2 - Keyboard Commands](#)

[4.5.3 - Pop Up Menu](#)

[4.5.4 - Toolbar](#)

[4.5.5 - 3D Display](#)

[5 - Associating components with footprints](#)

[5.1 - Principle](#)

[5.2 - Assignment](#)

[5.3 - Changing an existing assignment](#)

[5.4 - Filtering the footprint list:](#)

[6 - Automatic associations](#)

[6.1 - Equivalence files](#)

[6.2 - Format](#)

[6.3 - Association:](#)

[7 - Back- annotation file](#)

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.

Cvpcb.

2 - General characteristics

2.1 - Characteristics

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 files

- The netlist file *.net created by **Eeschema** (with or without footprint references)
- The auxiliary component assignment file *.cmp previously created by Cvpcb if exists.

2.3 - Output files

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).

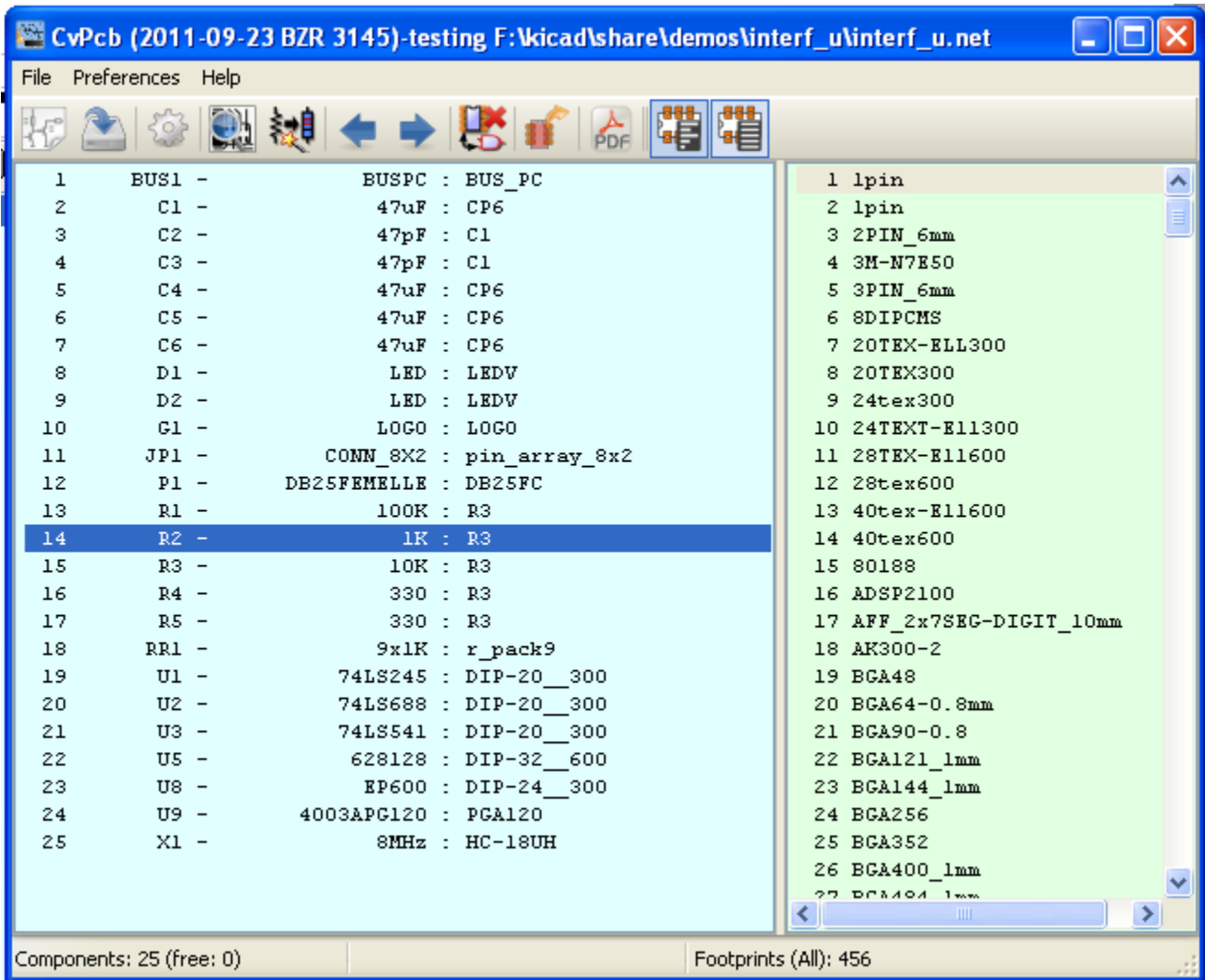
The standard extension of the file to be processed is **.net**.

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**.

4 - CVPCB Commands

4.1 - Main screen



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



Commands 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.

Cvpcb.

	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 authorized footprints for the current selected component.

4.3 - CVPCB Configuration

4.3.1 - General screen

Invoking the configuration menu displays the following screen:

Project file: F:\kicad\share\demos\interf_u\interf_u.pro

Footprint library files

- connect
- dip_sockets
- sockets
- discret
- pin_array
- divers
- libme

Add
Insert
Remove

Footprint alias files

- devcms

Add
Insert
Remove

Footprint documentation file

footprints_doc/footprints.pdf

Browse

User defined search paths

F:\kicad\share\modules

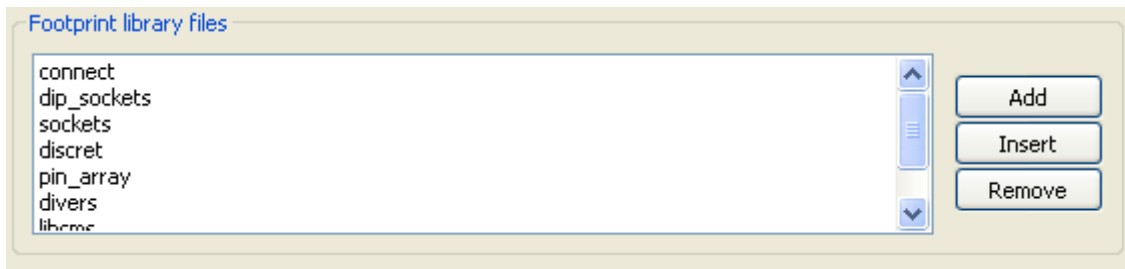
Add
Insert
Remove

Current search path list

- F:\kicad\share\demos\interf_u
- F:\kicad\share\modules
- F:\kicad\share\modules\packages3d
- F:\kicad\share\template

OK Cancel

4.3.2 - Footprint Library selection



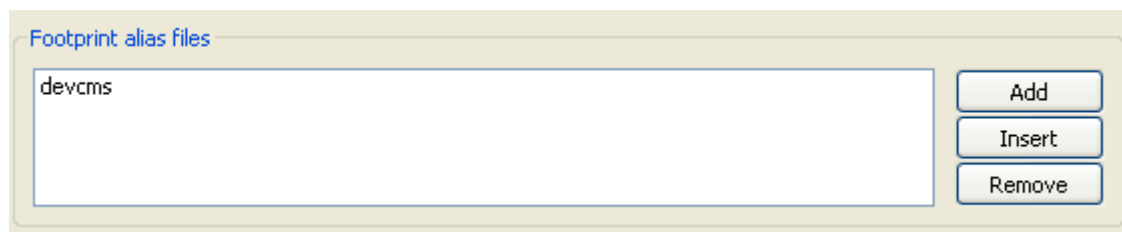
To select a file with the mouse:

- **Del** removes this name from the list.
- **Add** adds a new name to the list, to the end of the list.
- **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



To select with the mouse a file name.

- **Del** removes this name of the list.
- **Add** adds a new name to the list, to the end of the list.
- **Ins** adds a new name to the list, **before the** selected name

4.4 - Selecting default library path.

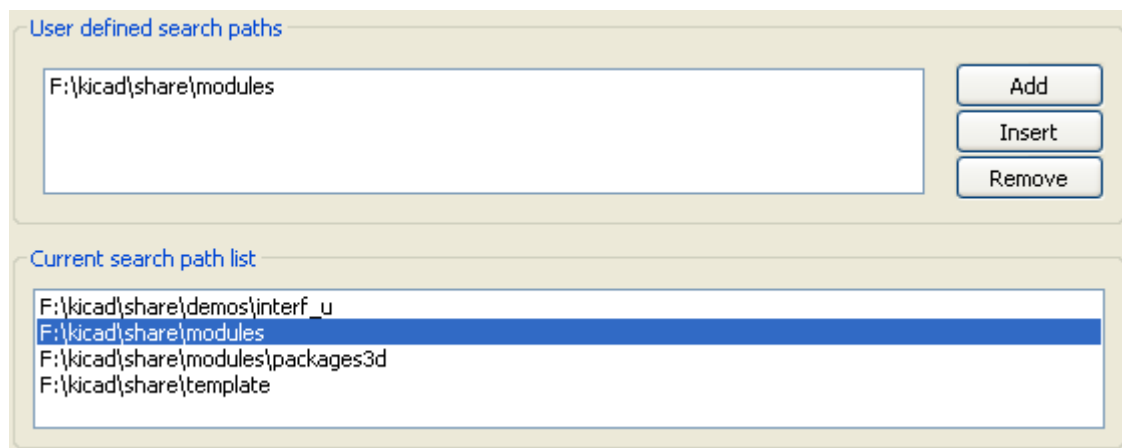
Default library paths are displayed by Cvpcb.

Cvpcb uses these paths to find the footprints libraries (.mod files) and the equivalence files (.equ files).

4.4.1 - Search paths:

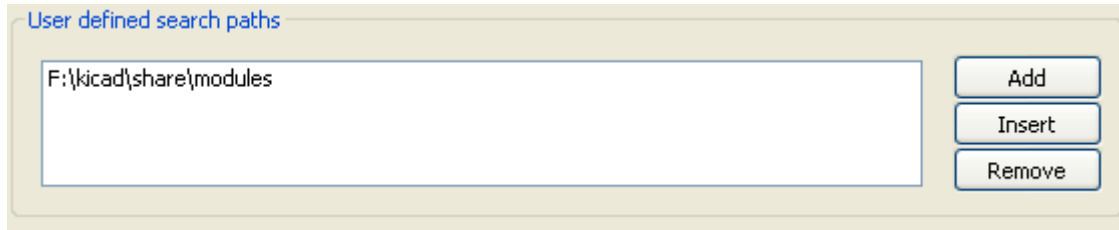
Cvpcb uses 2 types of paths:

- Paths automatically set by Cvpcb.
- Path added by users.



4.4.2 - Path added by users:

Cvpcb.



4.4.3 - Paths automatically set by Cvpcb

They depend on (partially) the D.O.S.

There is always **the working directory**.

Then:

- ***kicad/share/modules***.
- ***kicad/share/modules/packages3d*** (for 3D shapes files format **VRML** created par Wings3D).
- ***kicad/share/template***.

The root path kicad is

- The path where kicad binary is found (.../kicad/bin).

If not found:

Under Windows:

- c:\kicad
- d:\kicad

Under Unices:

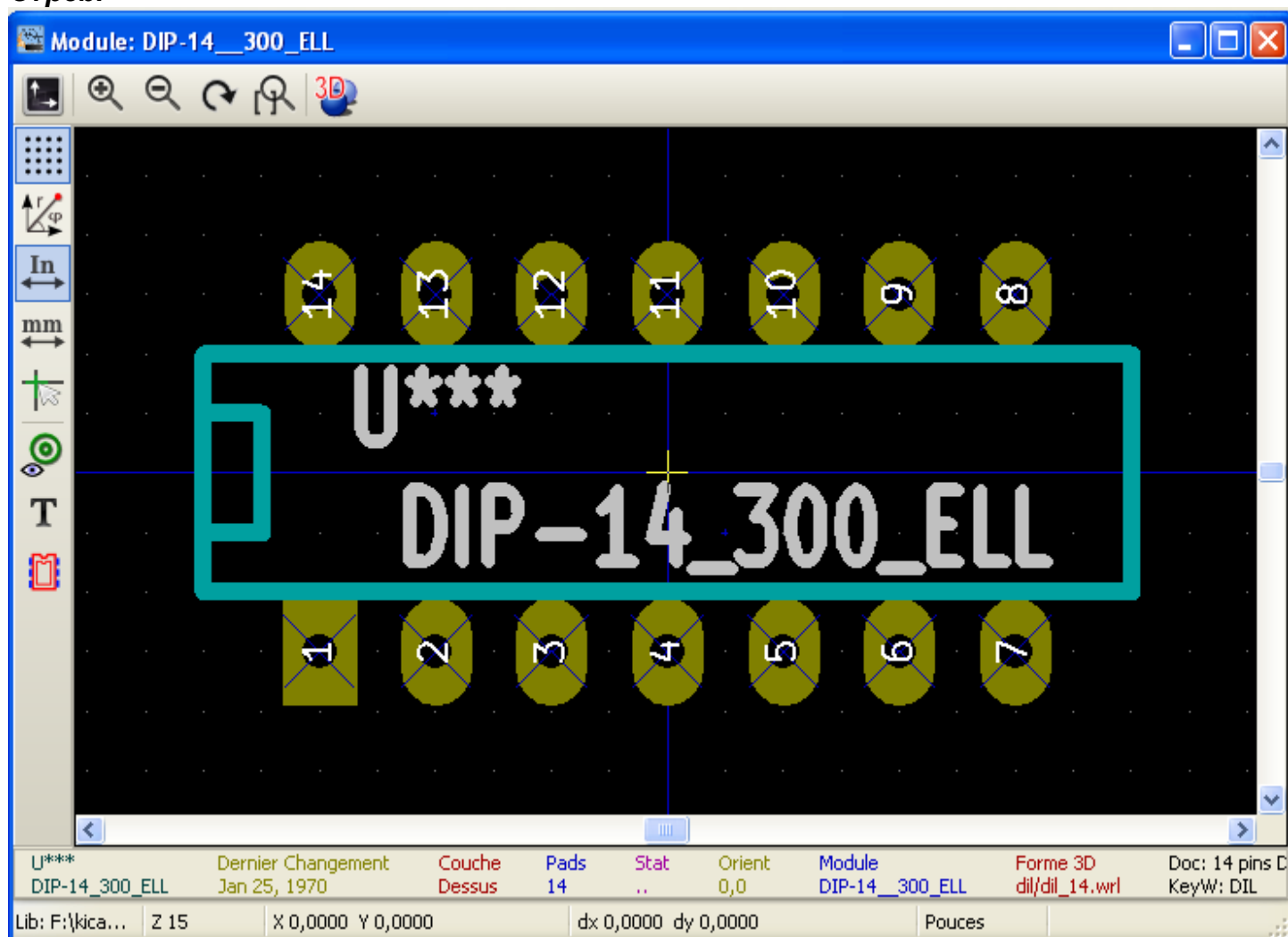
- /usr/local/kicad
- /usr/share/kicad

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)



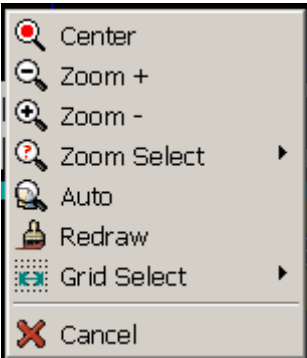
4.5.1 - Display

The coordinates 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.

4.5.3 - Pop Up Menu






Displayed by right-clicking the mouse:

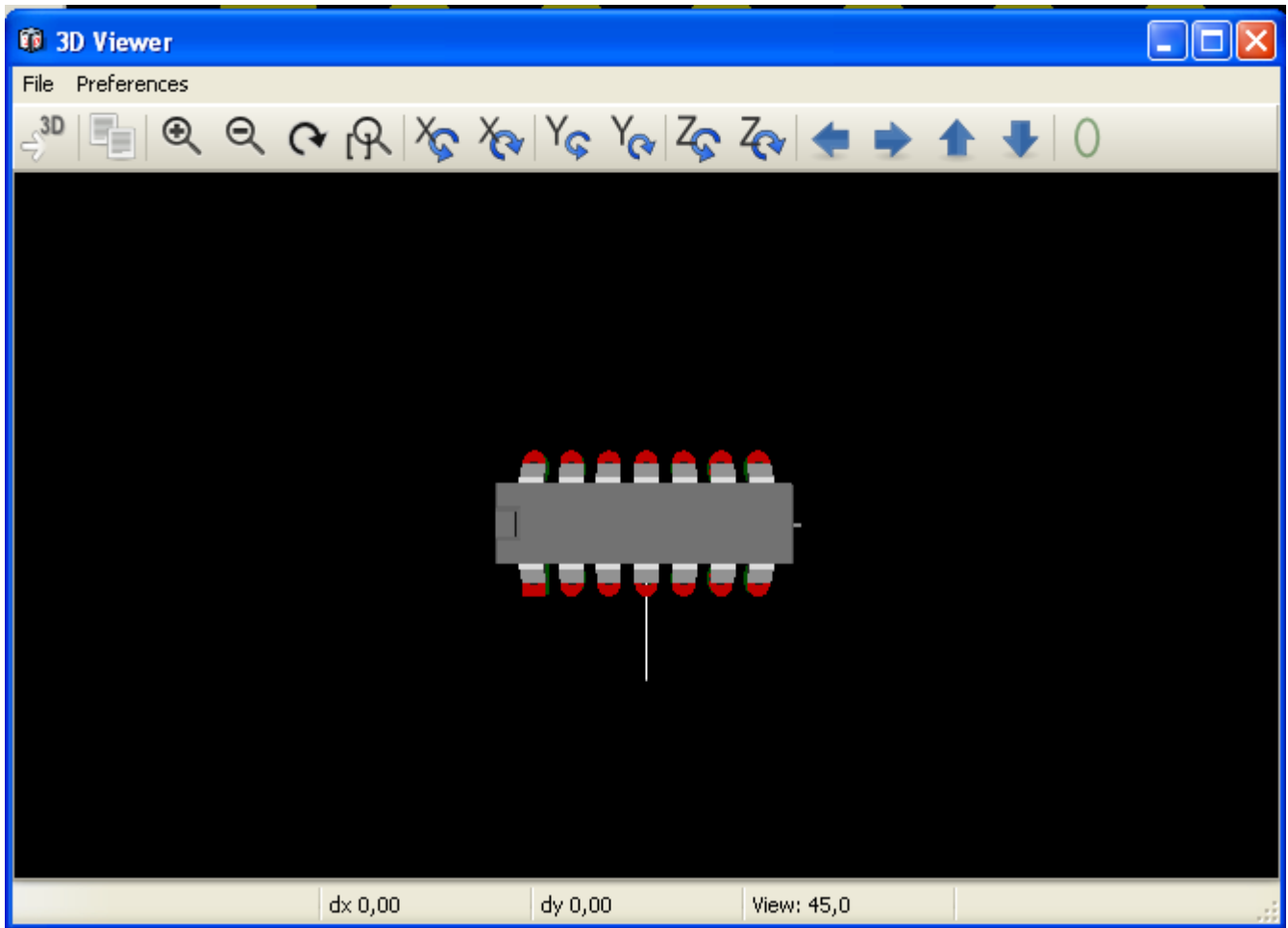
Zoom Selection (Select Zoom)	Direct selection of the display zoom .
Grid Selection (Grid Select)	Direct selection of the grid.

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:

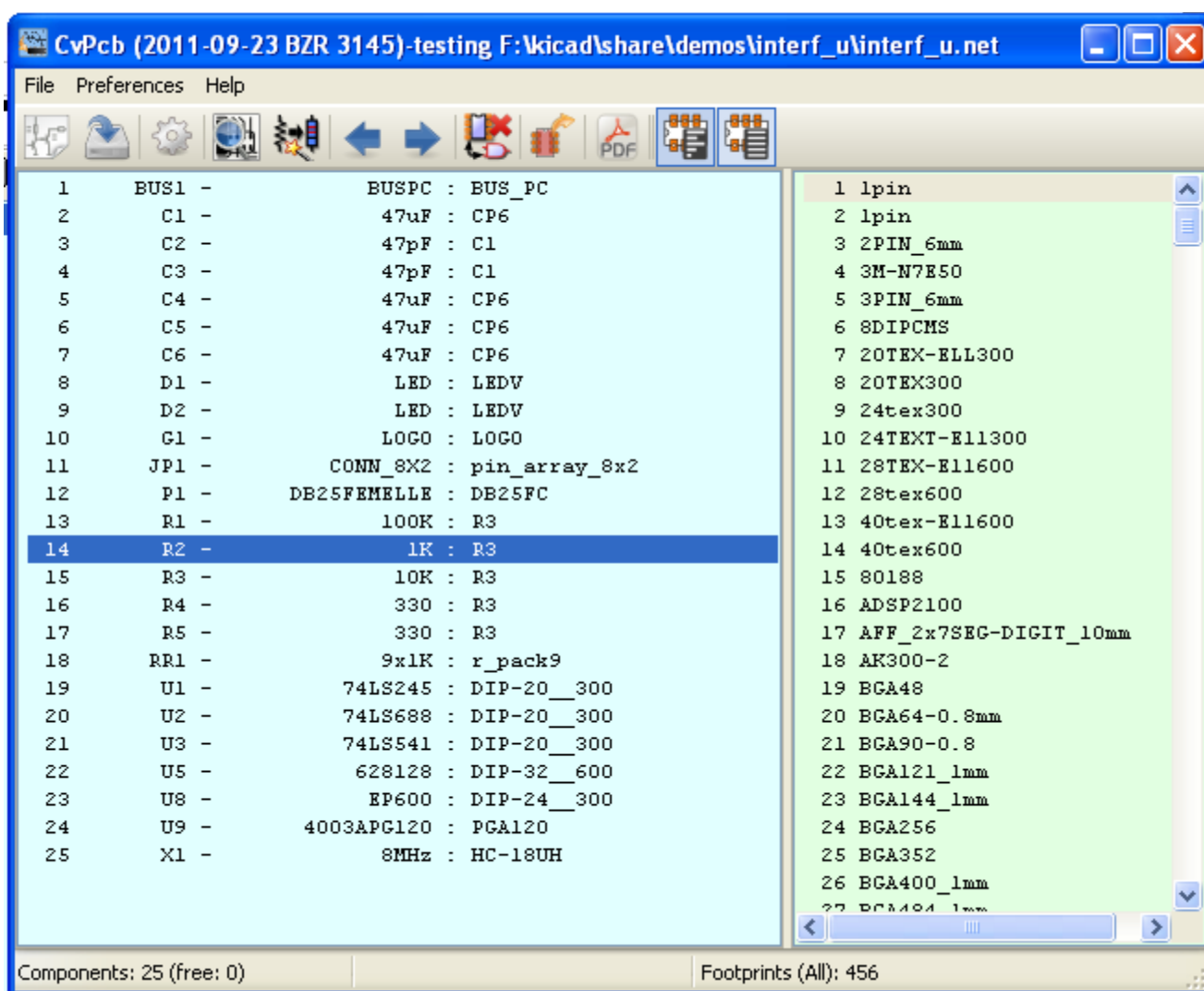
Double-click the **left mouse button** on the new desired **footprint**

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.

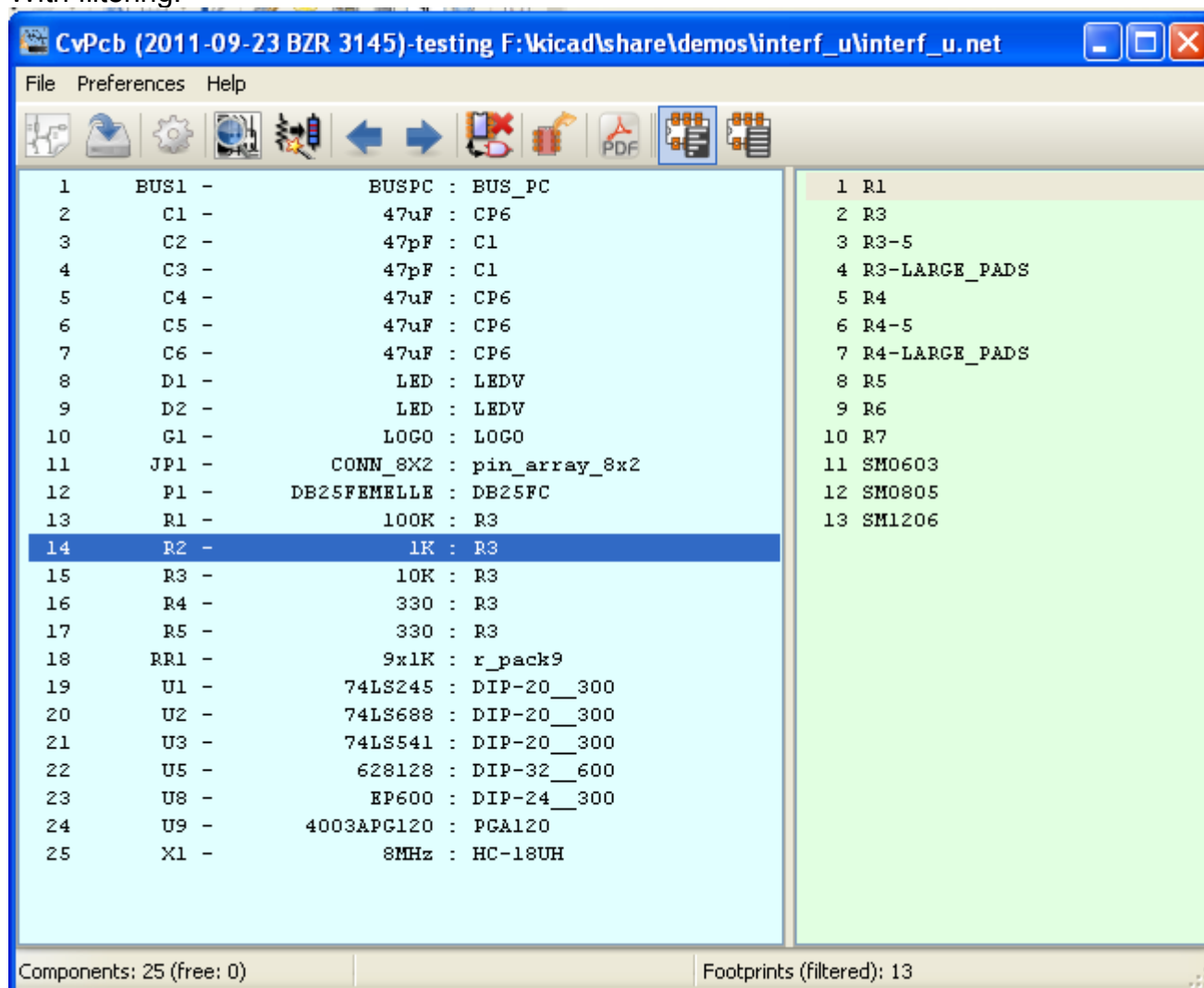
Cvpcb.

Without filtering:



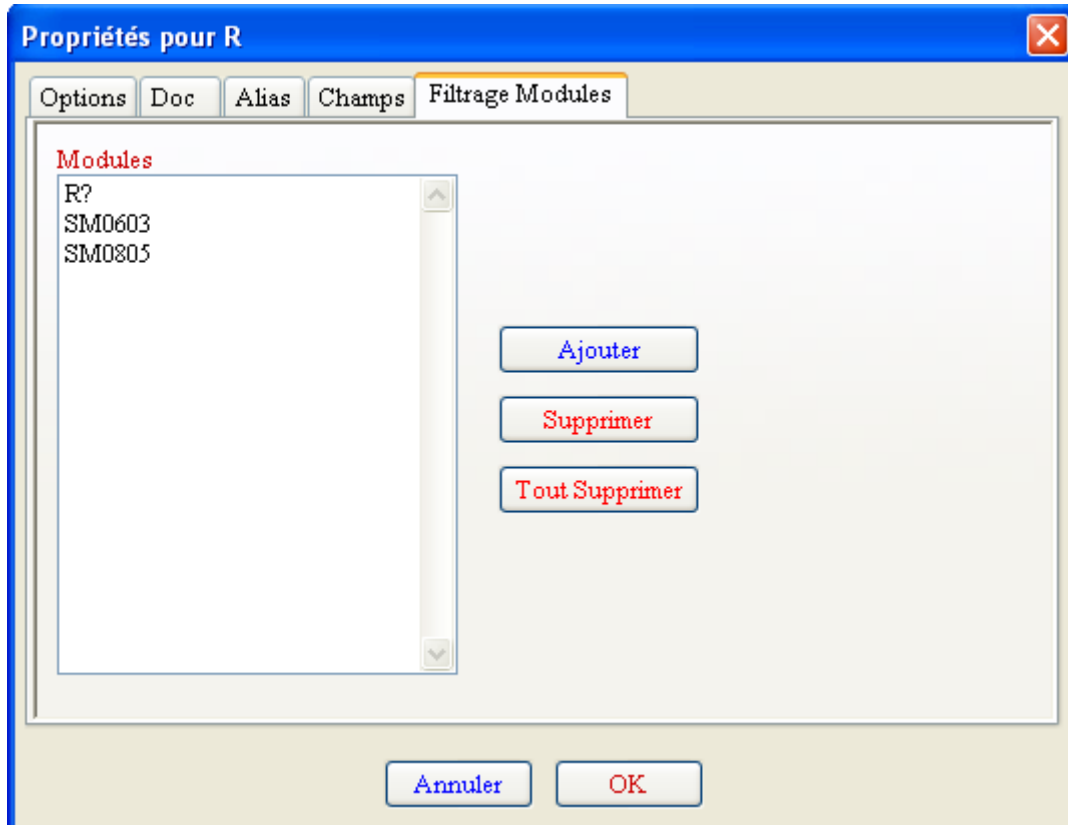
Cvpcb.



With filtering:



Under

Eeschema, the allowed footprint list was:



The icons   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'
```

Cvpcb.

```
'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'
```

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.