Cvpcb

Reference manual

Copyright

Contributors

Jean-Pierre Charras, Fabrizio Tappero.

Feedback

Please direct any comments or suggestions about this document to the kicad mailing list: *https://launchpad.net/~kicad-developers*

Acknowledgments

None

Publication date and software version

Published on September 30, 2011. Based on LibreOffice 3.3.2.

Note for Mac users

The kicad support for the Apple OS X operating system is experimental.

Table of Contents

1 - Introduction to Cvpcb

Cvpcb is a software tool that allows you to associate components in your schematic drawing to footprint components in the printed circuit board. This association is added to the netlist file.

Generally speaking, the netlist file 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 - Cvpcb features

Manual or automatic association

CVPCB allows for interactive assignment (manual) as well as automatic assignment via equivalence files. It is also possible to generate (if necessary) back-annotation files useful for the back assignment to the schematic.

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.

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. If necessary, extensions will be supplied by the defined in **cvpcb** configuration**.**

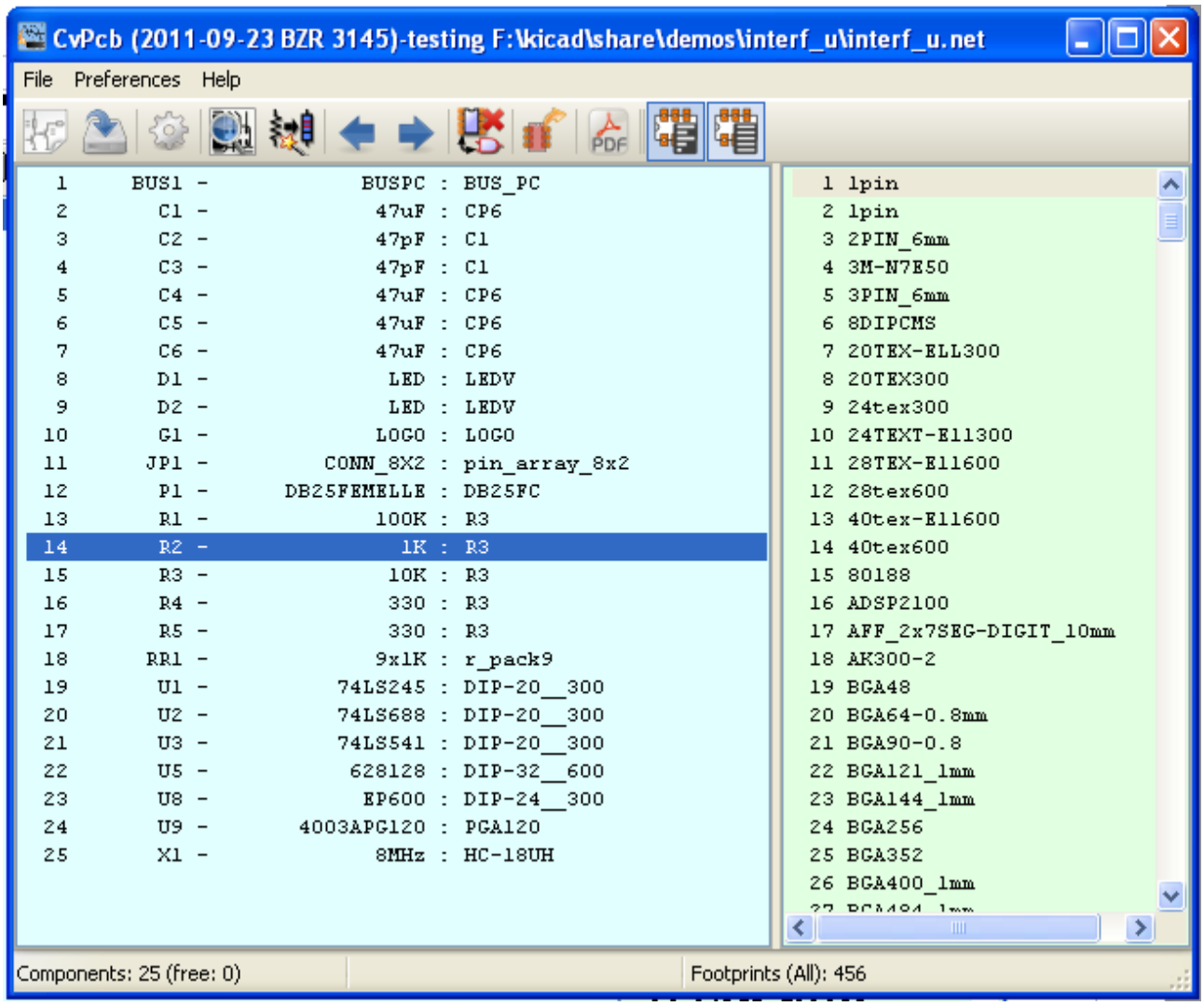
The two generated files will have the same name (with different extensions). The standard extension of the file to be processed is **.net**.

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

Main screen



The Component window on the left, displays the list of components appearing in the Netlist that has bean loaded.

The footprint window on the right, displays the list of footprints contained in the libraries that have been loaded.

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

Main top toolbar

The top toolbar allows for the following commands:

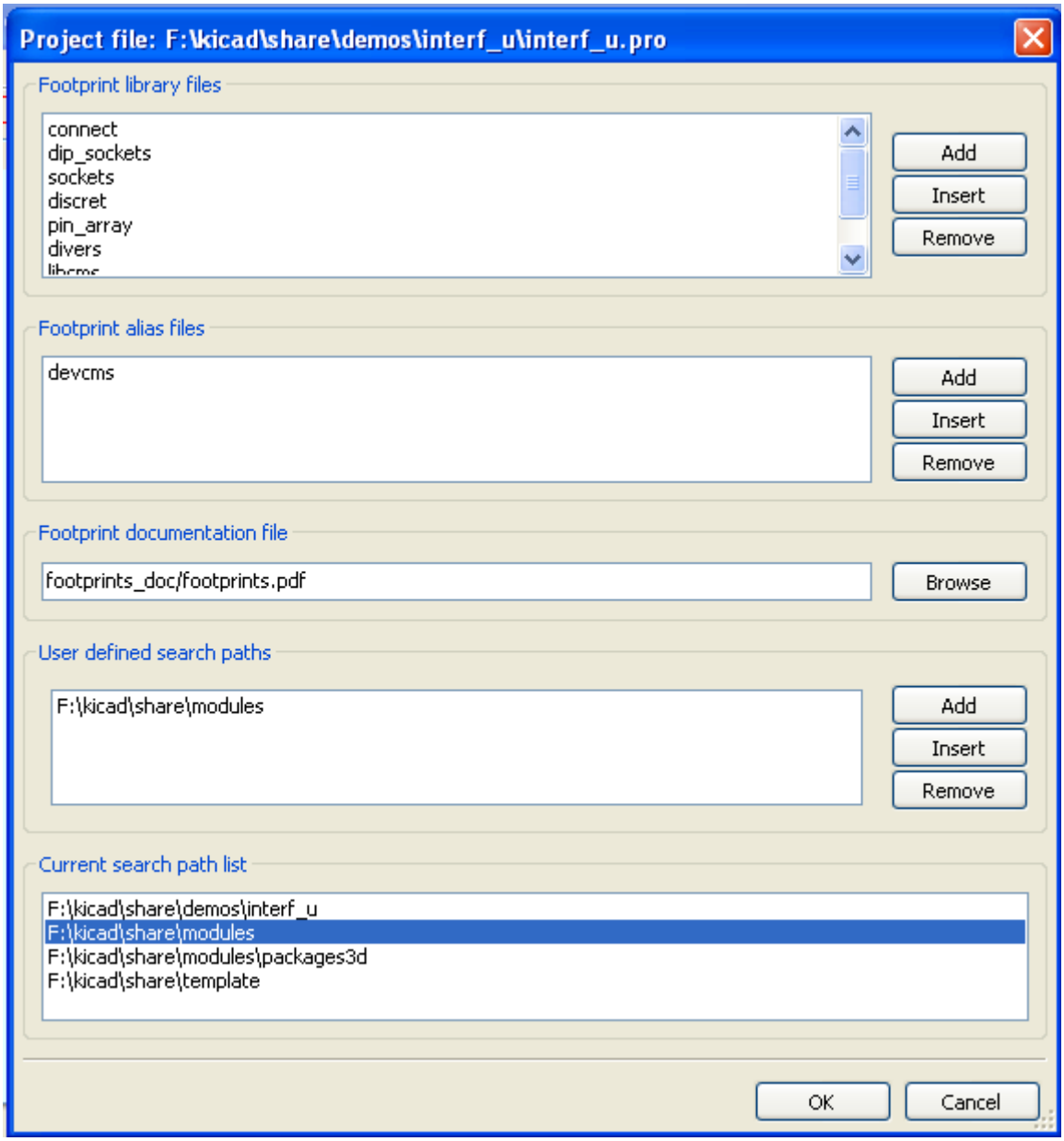
	Select the Netlist file to be processed.

	Save the assignment file .CMP and the updated Netlist .NET file.
	Invoke the Cvpcb configuration menu.
	Display the footprint of the component selected in the footprint window.
	Automatically associate components/footprints starting from the equivalence files. Using this order implies 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.
	Open the footprint documentation pdf file using the default pdf viewer.
	Enable or disable the footprint filtering to display the list of footprints.When the footprint filtering is enabled, the list of footprints shows only the “permitted” footprints for the current selected component.

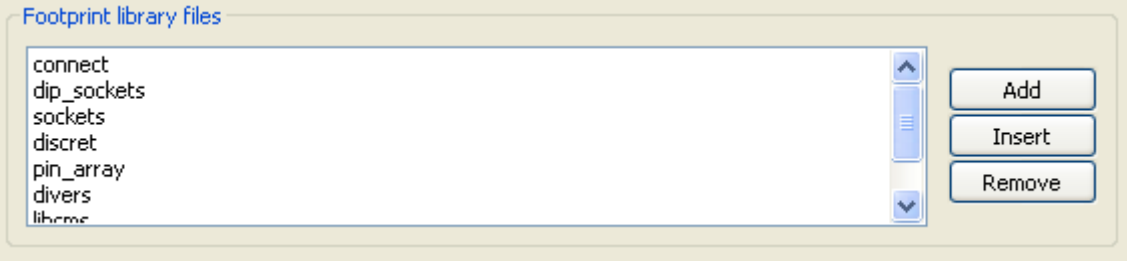
Cvpcb Configuration

Cvpcb configuration screen

Invoking the configuration menu displays the following screen:



Footprint library selection

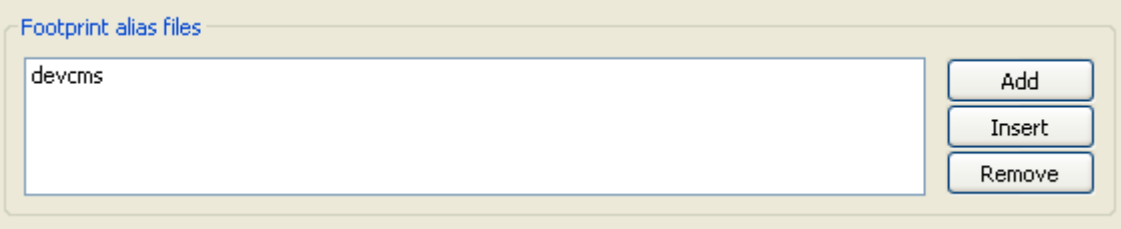


To select a file with the mouse:

- **Del:** removes this name from the list.
- **Add:** adds a new name 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.

Selecting 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

Selecting default library path.

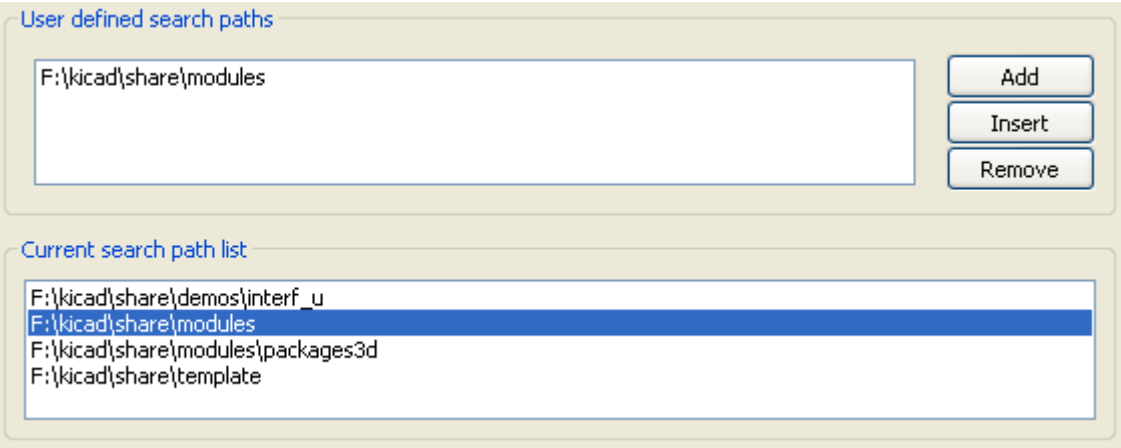
Default library paths are displayed by Cvpcb.

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

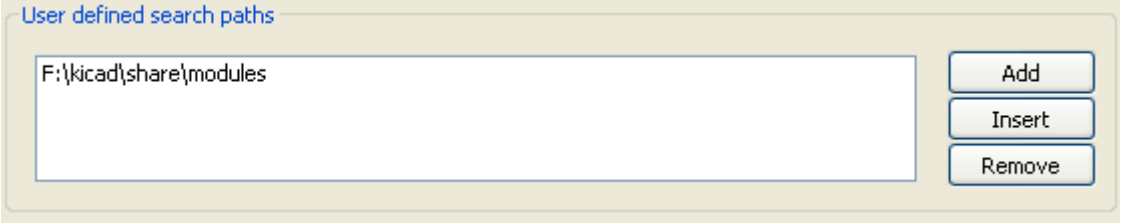
Search paths

Cvpcb uses 2 types of paths:

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



Path added by users



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 in which kicad is

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

If not found:

Under Windows:

- c:\kicad
- d:\kicad

Under Linux:

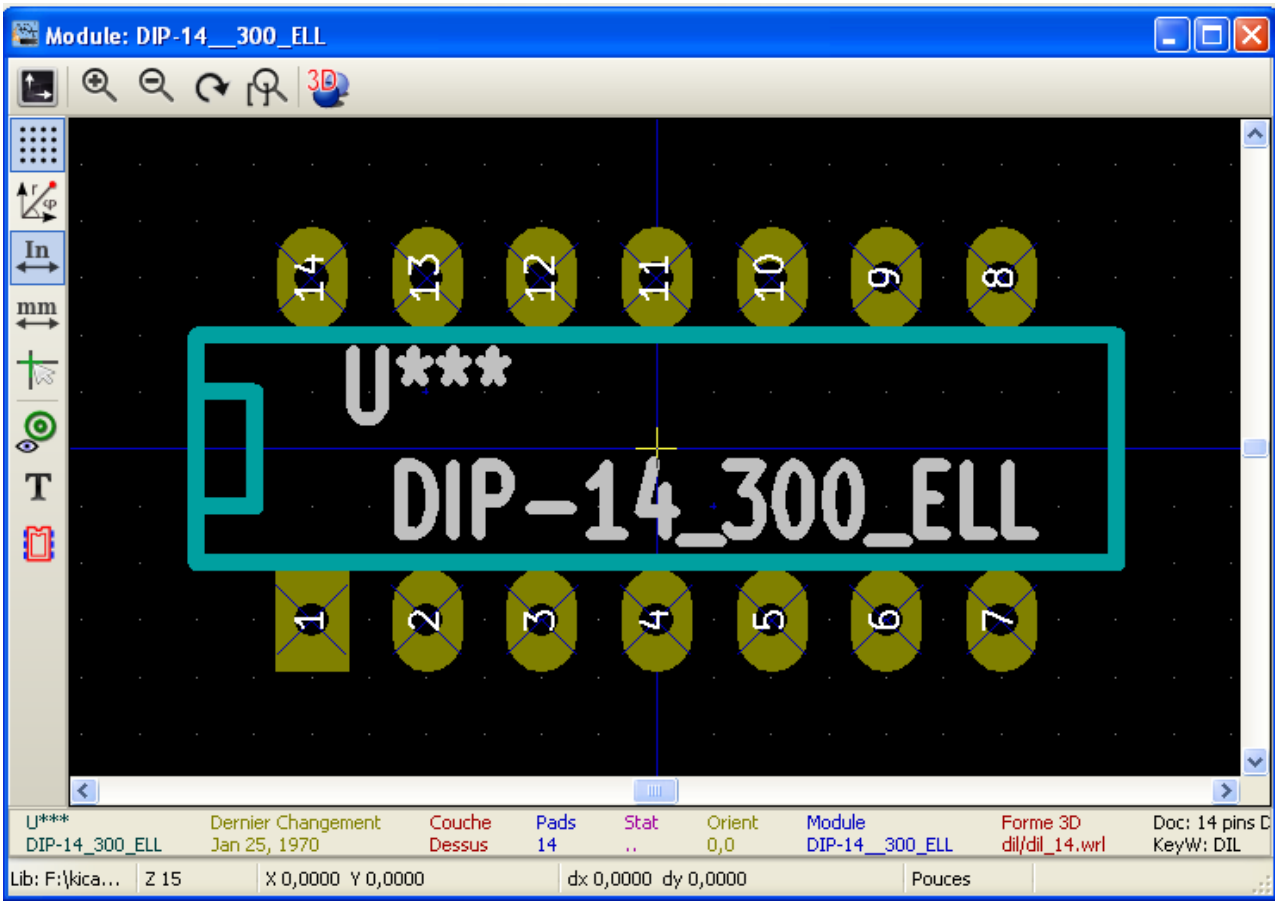
- /usr/local/kicad
- /usr/share/kicad

Viewing the current footprint

The View command allows to display 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.



Additional information

The coordinates of the cursor are displayed at the bottom of the screen:

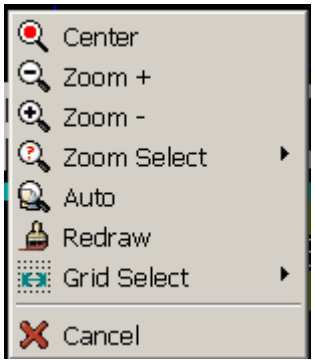
Absolute coordinates (X nnnn Y nnnn) and relative coordinates (dx nnnn dy nnnn)

The relative coordinates are set to zero by the space bar.

Keyboard commands

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

Right-click menu



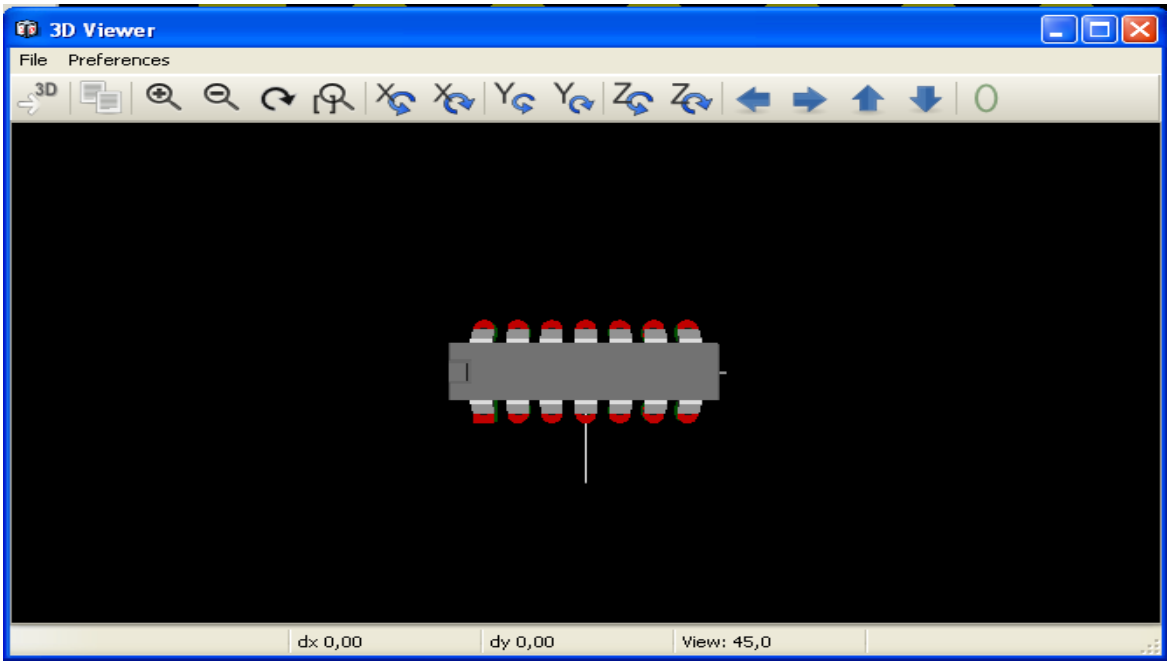
Displayed by right-clicking the mouse:

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

Toolbar

	Display options
	Zoom levels
	Display 3D object

3D Display



5 - Associating components with footprints

How it works

In the footprint window double-click on the name of the desired footprint (this name will be highlighted). To assign it to 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.

Assignment

Double-click the left mouse button on the desired footprint

Changing an existing assignment

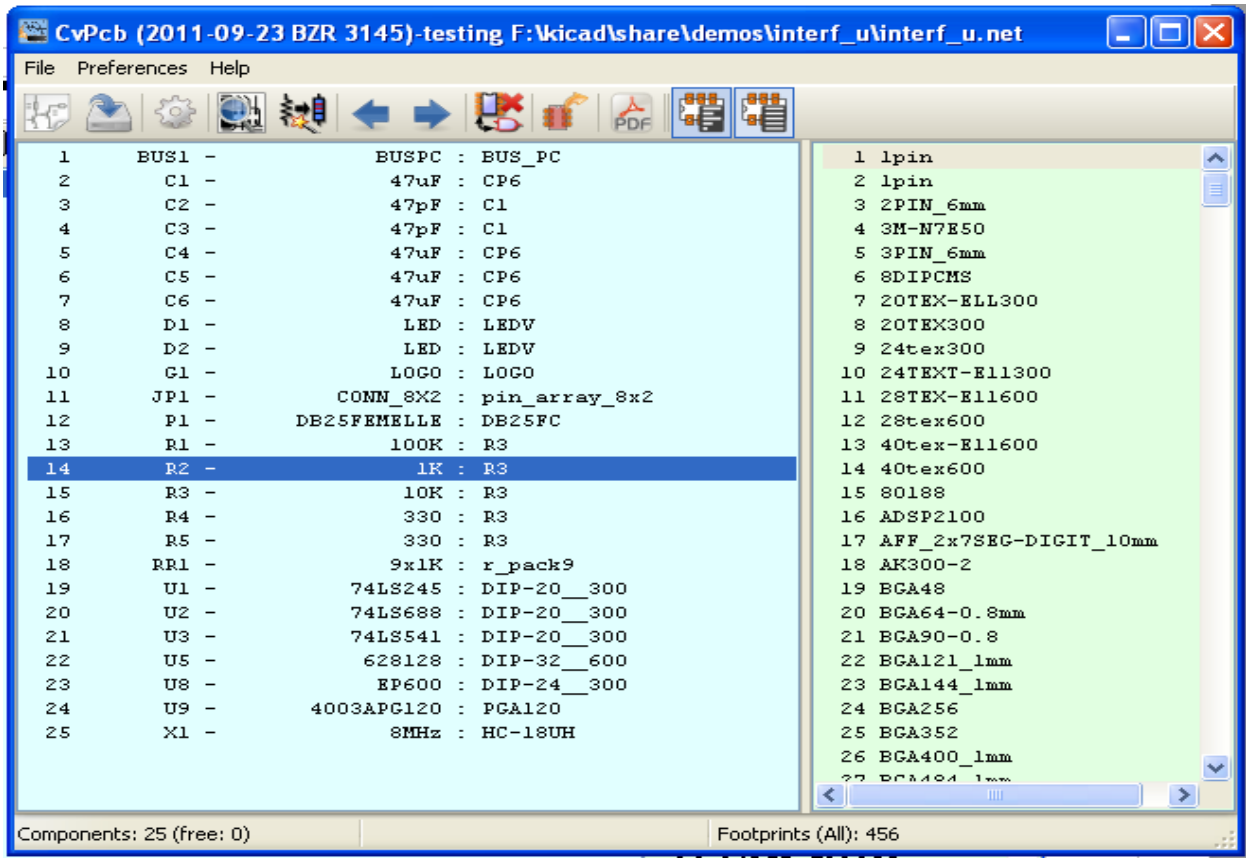
This is done like a new assignment:

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

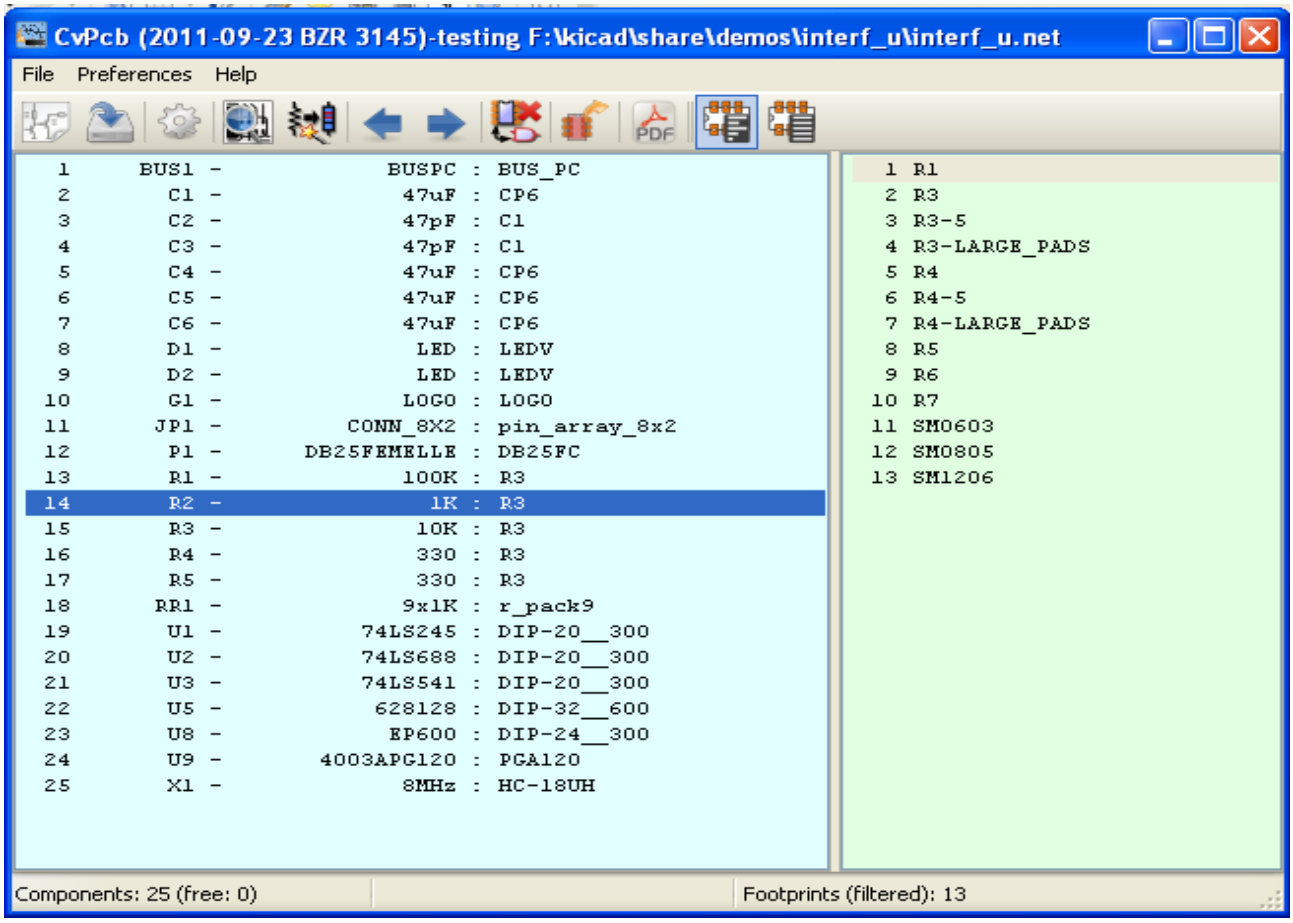
Filtering the footprint list

If the selected component is highlighted when the filter option is enabled, the displayed footprint list in Cvpcb is filtered accordingly.

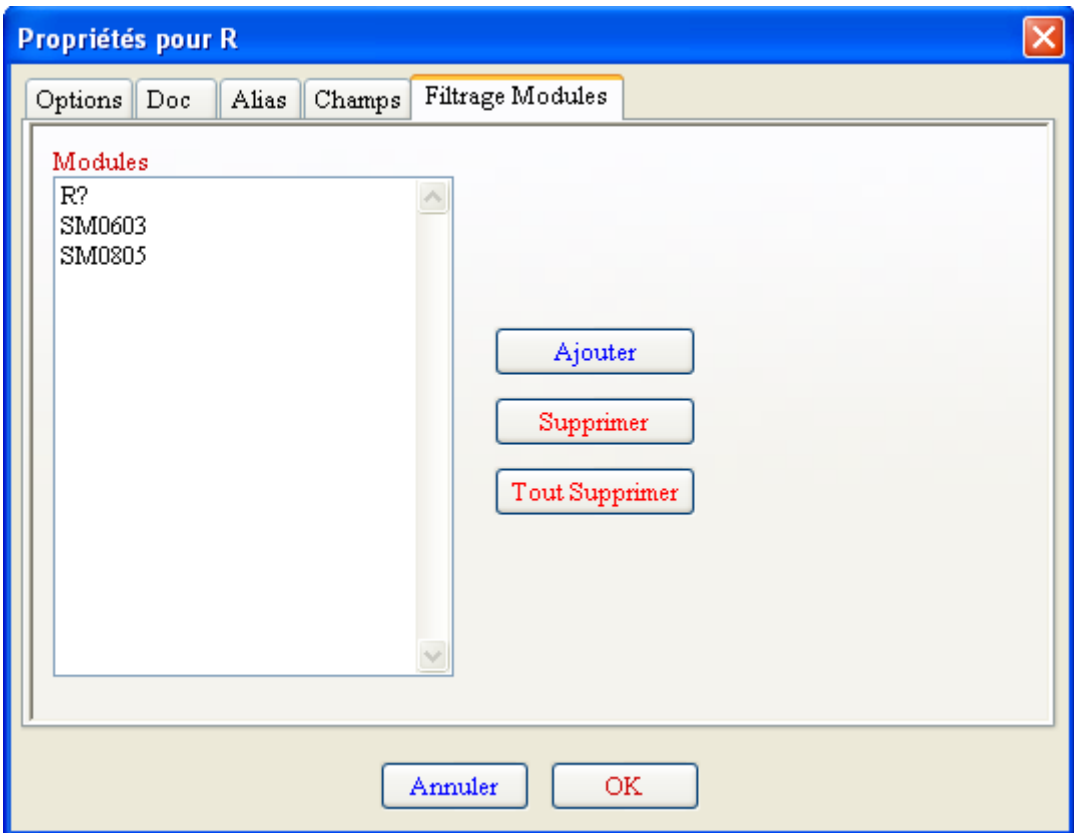
Without filtering:



With filtering:



Under Eeschema, the allowed footprint list was:



The icons  enable and disable the filtering feature. When the filtering is not enabled, the full footprint list is shown.

6 - Automatic associations

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, dip packages or other criterias)

Refer to the section “Selecting the equivalence files” for more information.

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 you can see an example:

```
#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'
```

Automatic component association

The automatic association process is enabled by the icon  .

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 the 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.