CvPcb

Reference manual

Copyright

Contributors

Jean-Pierre Charras, Fabrizio Tappero, Wayne Stambaugh.

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 March 2, 2012. 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 tool that allows you to associate components in your schematic to component footprints used when laying out the printed circuit board. This association is added to the net list file created by the schematic capture program Eeschema.

Typically the net list file generated by Eeschema does not specify which printed circuit board footprint is associated with each component in the schematic. Although this is not always the case as component footprints can be associated during schematic capture by setting the component's footprint field. CvPcb provides a convenient method of associating footprints to components. It

provided footprint list filtering, footprint viewing, and 3D component model viewing to help ensure the correct footprint is associated to each component.

Components can be assigned to their corresponding footprints manually or automatically by creating equivalence files. Equivalence files are look up tables associating each component with it's footprint.

This interactive approach is simpler and less error prone than directly associating the footprints in the schematic editor because as well as allowing for automatic association, CvPcb allows you to see the list of available footprints available and display them on the screen to ensure you are associating the correct footprint.

2 - CvPcb Features

2.1 - Manual or Automatic Association

CvPcb allows for interactive assignment (manual) as well as automatic assignment via equivalence files. It is also possible to generate back-annotation files useful for automatically associating the footprints selected by CvPcb schematic captured by Eeschema.

2.2 - Input Files

- The net list file (*.net) created by Eeschema with or without footprint associations.
- The auxiliary component assignment file (*.cmp) previously created by CvPcb if one exists.

2.3 - Output Files

Two files are generated for Pcbnew:

- The net list file with footprint associations.
- An auxiliary component association file (*.cmp).

3 - Invoking CvPcb

CvPcb is typically invoked from the schematic capture program Eeschema. Eeschema automatically passes the correct net list file name to CvPcb. Before running CvPcb for the first time for each project, you must first save the initial net list by selecting the "Generate Netlist" entry in to "Tools" menu or click on the generate netlist button on the top tool bar in Eeschema. By default the net list file has the same name as the project with a "net" file extension. If the net list file for the project already exists, all footprint associations will be preserved. After the initial net list file is created by Eeschema, CvPcb can be invoked directly from the KiCad project manager. CvPcb can also be invoked as a stand alone program rather than being launch from the KiCad project manager or the schematic editor. If CvPcb is run as a stand alone program, the net list file must be opened manually by selecting the "Open" entry in the "File" menu or clicking the "Open" file button on the tool bar.

4 - CvPcb Commands

4.1 - Main Screen
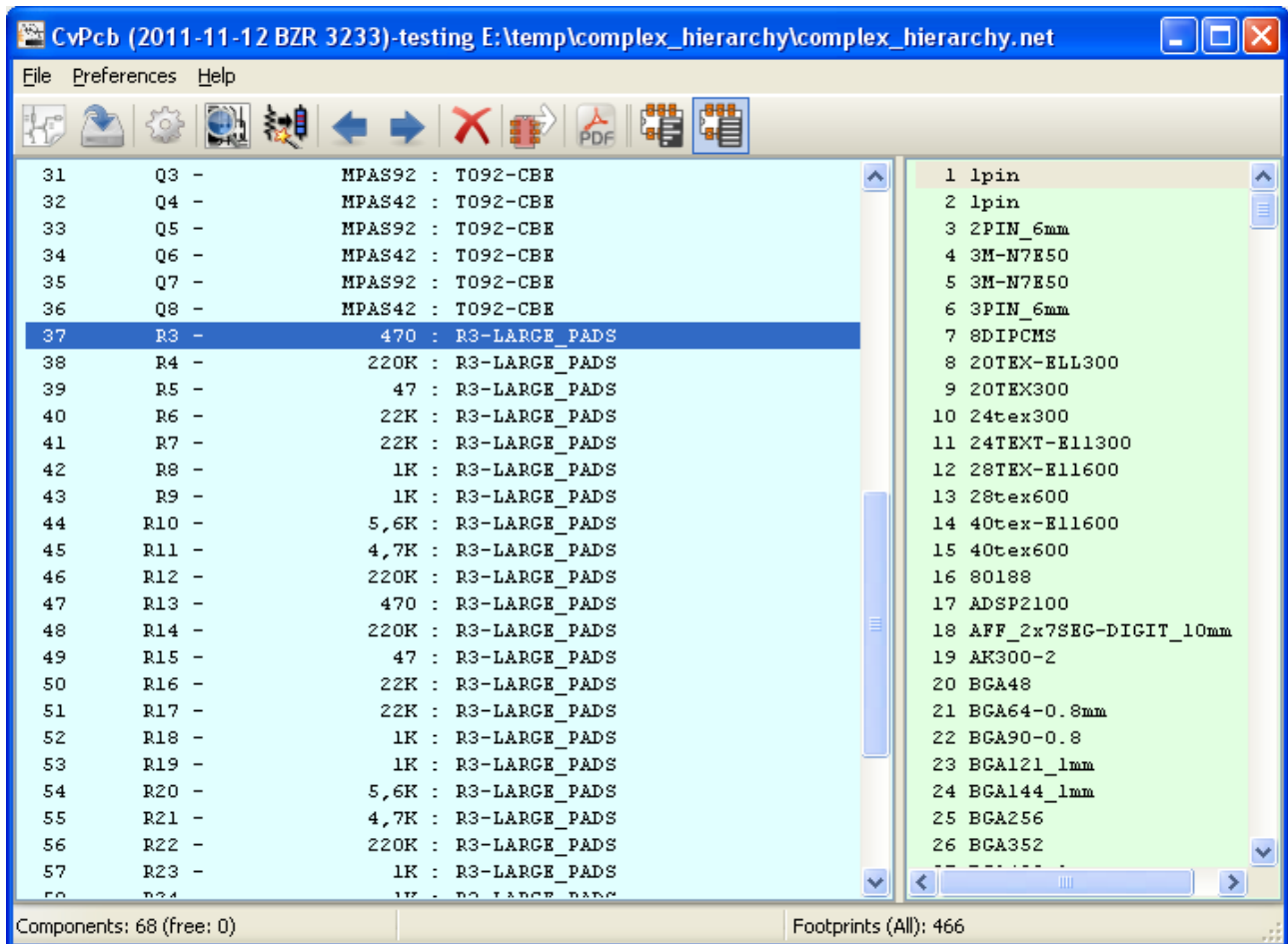
The component window on the left, displays the list of components appearing in the net list file that has bean loaded. The footprint window on the right, displays the list of footprints contained in the libraries that have been loaded. The component window will be empty if no file is loaded and the footprint window can be also empty if no footprint libraries are found.

4.2 - Main Window Toolbar.

The top toolbar allows for easy access to the following commands:

	Select the net list file to be processed.
	Save the footprint association file (.cmp) and the updated net list (.net) file.
	Invoke the CvPcb configuration menu.
	Display the footprint of the component selected in the footprint window.
	Automatically associate footprints with components starting using an equivalence file.
	Automatically select the previous component in the list without a footprint association.
	Automatically select the next component in the list without a footprint association.

	Delete all footprint assignments.
	Generate footprint assignment back annotation file.
	Open the selected footprint documentation pdf file using the default pdf viewer.
	Enable the footprint filtering to limit the list of footprints for the selected component.
	Disable the footprint filtering to limit the list of footprints for the selected component.

4.3 - Main Window Keyboard Commands

The following table lists the keyboard commands for the main window:

|	
Right Arrow	Activate the footprint pane if the component pane is currently activated.
Left Arrow	Activate the component pane if the footprint pane is currently activated.
Up Arrow	Select the previous item of the currently selected list.
Down Arrow	Select the next item of the currently selected list.
Page Up	Select the item up one full page of the currently selected list.
Page Down	Select the item down one full page of the currently selected list.
Home	Select the first item of the currently selected list.
End	Select the last item of the currently selected list.

4.4 - CvPcb Configuration

4.4.1 - CvPcb Configuration Screen

Invoking the "Libraries" entry in the "Preferences" menu displays the library configuration dialog shown below.
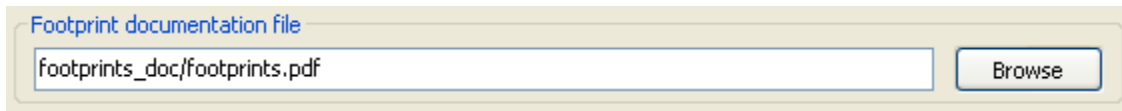
4.4.2 - Selecting Footprint Libraries

This section of the footprint library configuration dialog is used to add, remove, and change the search order of the footprint libraries for the current project. The library order is critical when searching for footprints with duplicate names. CvPcb will use the first occurrence of the footprint name it finds. If you create a new footprint, it is always a good idea to give it a unique name to prevent naming conflicts. This is a known issue and will be fixed in a future version of KiCad. Please note that changing these libraries will also effect Pcbnew.
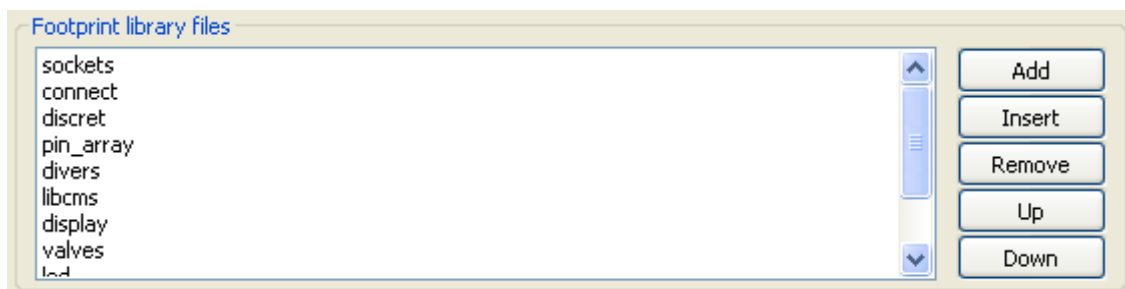
- **Remove:** Removes the selected footprint library from the list.
- **Add:** Adds a new footprint library to the end of the list.
- **Insert:** Inserts a new footprint library to the list before the currently selected library.
- **Up:** Move the currently select library up the list.
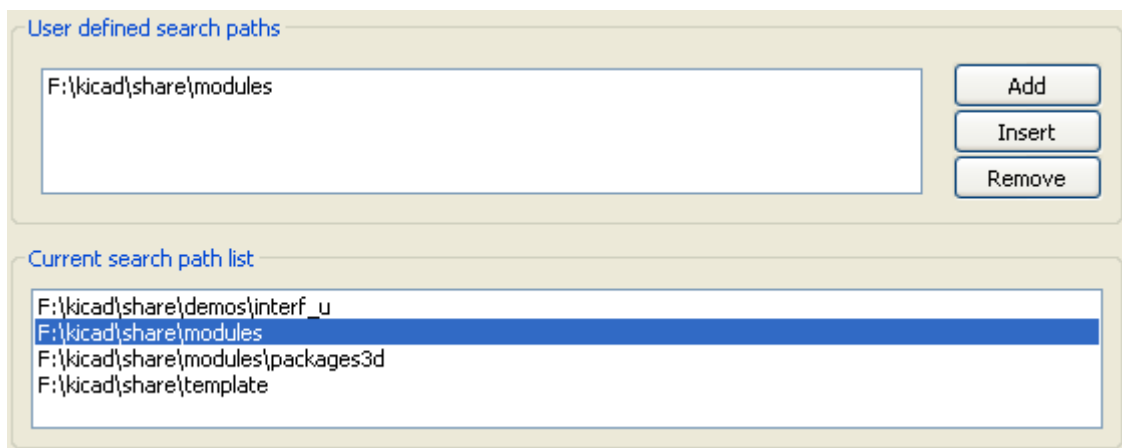- **Down:** Move the currently selected library down the list.

4.4.3 - Changing the Footprint Documentation File.

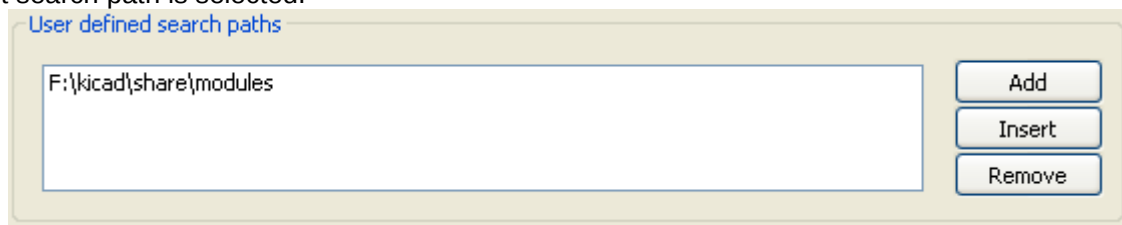Select the "Browse" button to select a new footprint documentation file with the display file select dialog.

4.5 - Changing Footprint Library Search Paths.

CvPcb uses two types of paths: the default paths automatically set by KiCad when a new project is created and paths added by the user. These paths are used to find the footprints library files (.mod), equivalence files (.equ), and 3D model files (.wrl) used by CvPcb. The default paths cannot be edited. Only new user defined paths can be added to the search path list.

Changing User Define Paths.

Click the "Add" button to add a new path after the selected entry in the search path list. Click the "Insert" button to insert a new path before the selected entry in the search path list. Click the "Remove" button to remove the selected user defined search path. Clicking the "Remove" button will have no effect if a default search path is selected.

4.5.1 - Default Library Paths

By default CvPcb internally uses a set of predefined paths used to search for footprint libraries. These paths are operating system dependent. It is generally preferable to use relative paths

rather than absolute paths whenever possible to prevent platform dependency problems. In other words: "*c:\Program* Files\kicad\share\" has no meaning and will fail on Linux and OSX.

The default Linux library paths will be as follows:

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

Where the root path is relative to the binary path where KiCad is installed. Typically on Linux Kicad is installed in the /usr/bin path. Therefore the root path would be /usr.

4.6 - Viewing the Current Footprint

The view footprint command displays the footprint currently selected in the *footprint* window. A 3D model of the component can be shown if it has been created and assigned to the footprint. Below is the footprint viewer window.
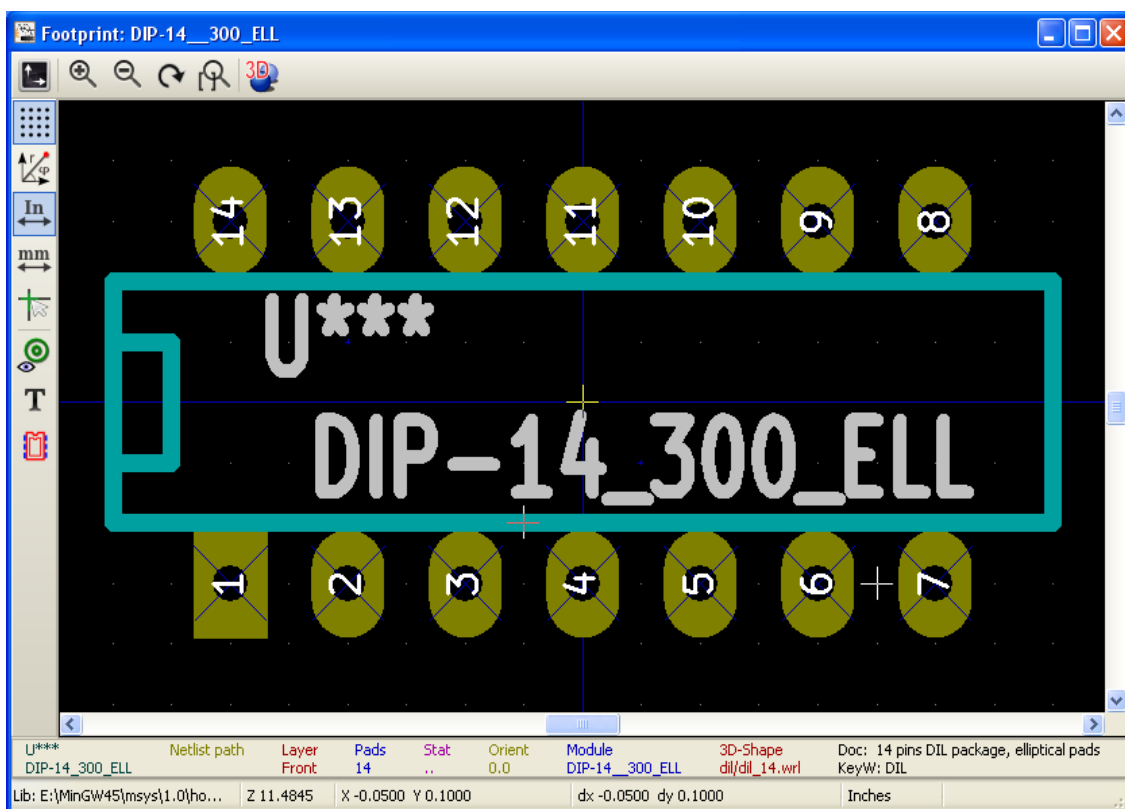
4.6.1 - Status Bar Information

The status bar is located a the bottom of the CvPcb new main window and provides useful information to the user. The following table defined the contents of each pane in the status bar.

Pane	Description
1	Current command help information
2	The current zoom level
3	The absolute position of the cursor in the current units and notation
4	The relative position of the cursor in the current units and notation

5	The current position units

### *4.6.2 - Keyboard Commands*

F1	Zoom In
F2	Zoom Out
F3	Refresh Display
F4	Move cursor to center of display window
Home	Fit footprint into display window
Space Bar	Set relative coordinates to the current cursor position
Right Arrow	Move cursor right one grid position
Left Arrow	Move cursor left one grid position
Up Arrow	Move cursor up one grid position
Down Arrow	Move cursor down one grid position

4.6.3 - Mouse Commands

Scroll Wheel	Zoom in and out at the current cursor position
Ctrl + Scroll Wheel	Pan right and left
Shift + Scroll Wheel	Pan up and down
Right Button Click	Open context menu

4.6.4 - Context Menu
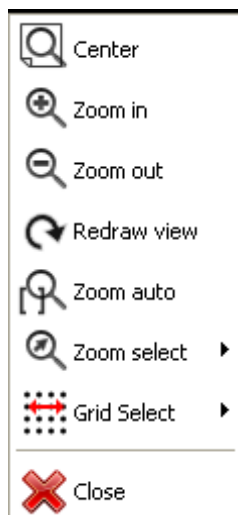
Displayed by right-clicking the mouse:

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

4.6.5 - Horizontal Toolbar

	Show display options dialog
	Zoom in

	Show display options dialog
	Zoom out
	Redraw
	Fit drawing in display area
	Open 3D model viewer

4.6.6 - Vertical Toolbar

	Show or hide the grid
	Show coordinates in polar or rectangular notation
	Display coordinates in inches
	Display coordinates in millimeters
	Toggle cursor style
	Toggle between drawing pads in sketch or normal mode
	Toggle between drawing text in sketch or normal mode
	Toggle between drawing edges in sketch or normal mode

4.7 - Viewing the Current 3D Model
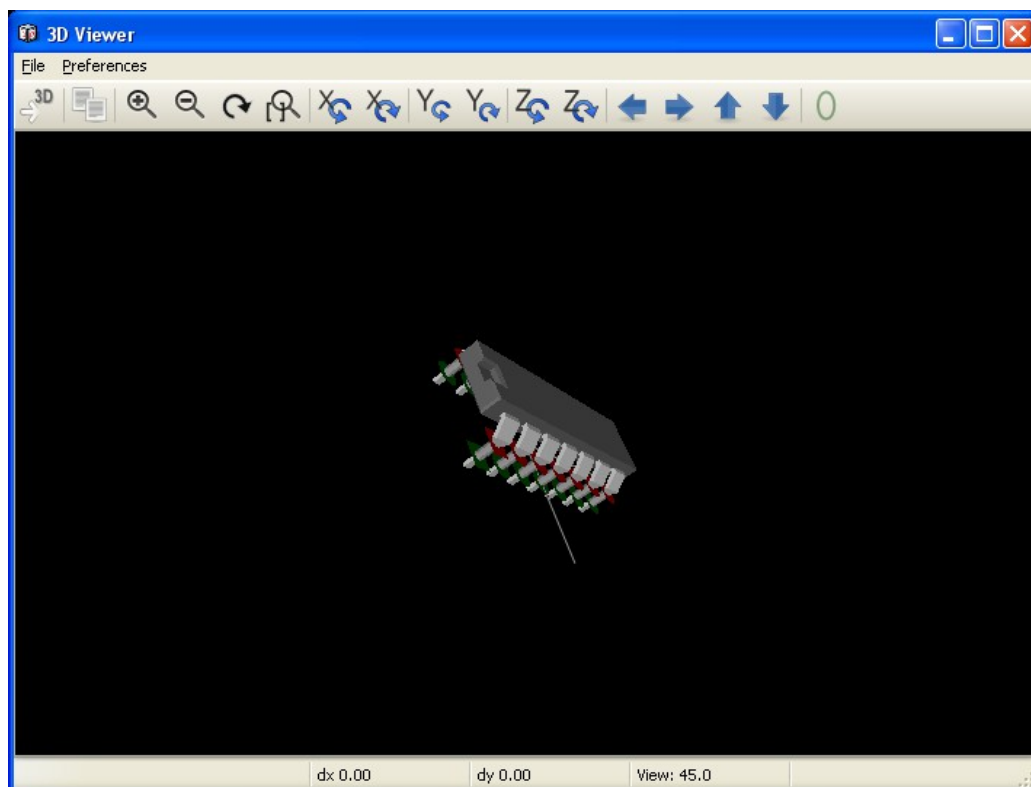
4.7.1 - Mouse Commands

Scroll Wheel	Zoom in and out at the current cursor position
Ctrl + Scroll Wheel	Pan right and left
Shift + Scroll Wheel	Pan up and down

4.7.2 - Horizontal Toolbar

	Reload the 3D model
	Copy 3D image to clipboard
	Zoom in
	Zoom out
	Redraw
	Fit drawing in display area
	Rotate backward along the X axis
	Rotate forward along the X axis
	Rotate backward along the Y axis
	Rotate forward along the Y axis
	Rotate backward along the Z axis
	Rotate forward along the Z axis
	Pan left
	Pan right
	Pan up
	Pan down
	Toggle orthographic projection mode on and off

5 - Using CvPcb to Associate Components with Footprints

5.1 - Manually Associating Footprints with Components

To manually associate a footprint with a component first select a component in the component pane. Then select a footprint in the footprint pane by double-clicking the left mouse button on the name of the desired footprint. The unassigned next component in the list is automatically selected. Changing the component footprint is performed in the same manner.

5.2 - 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.
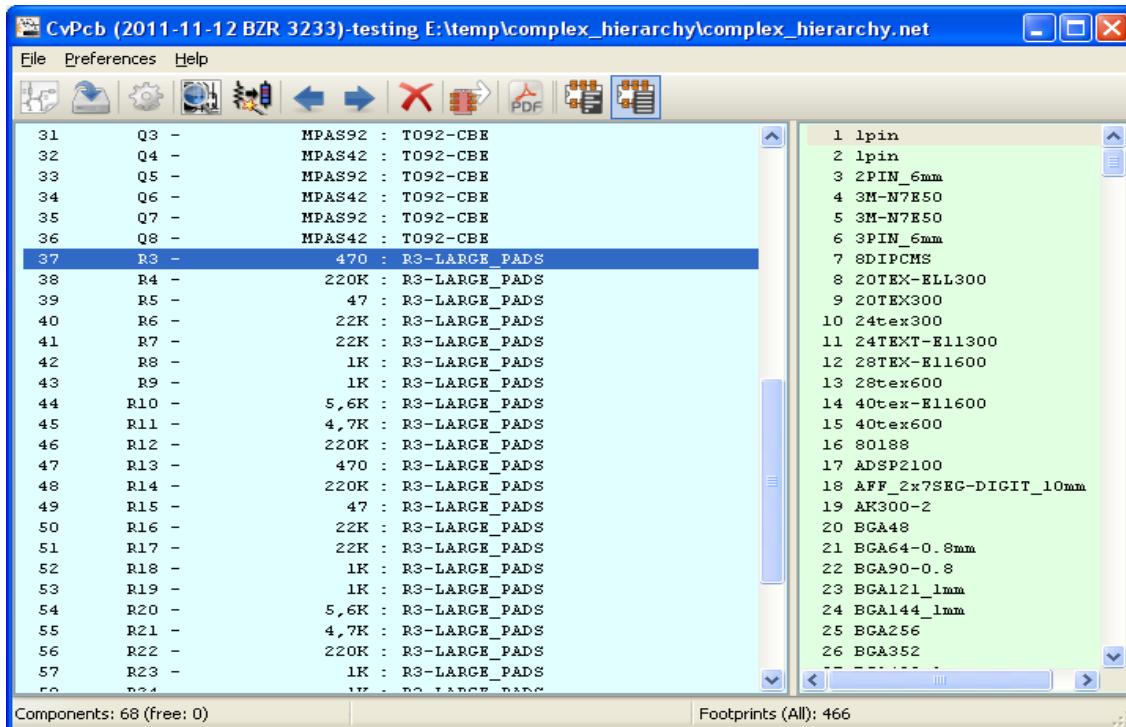
With filtering.

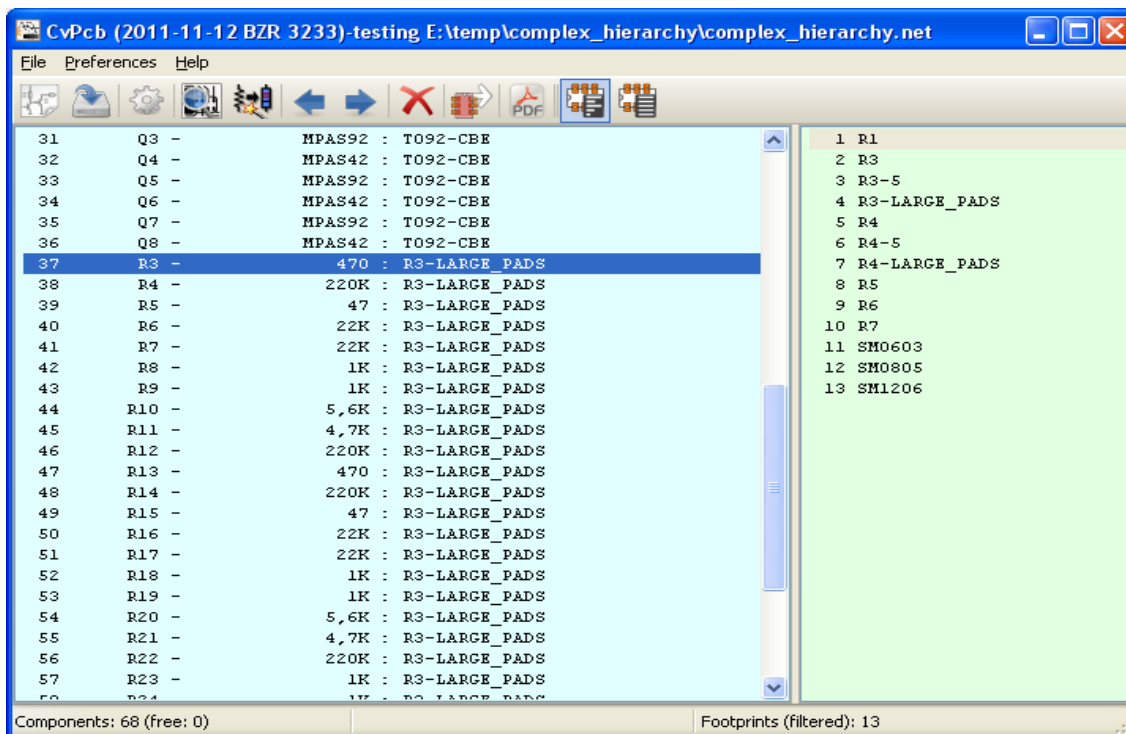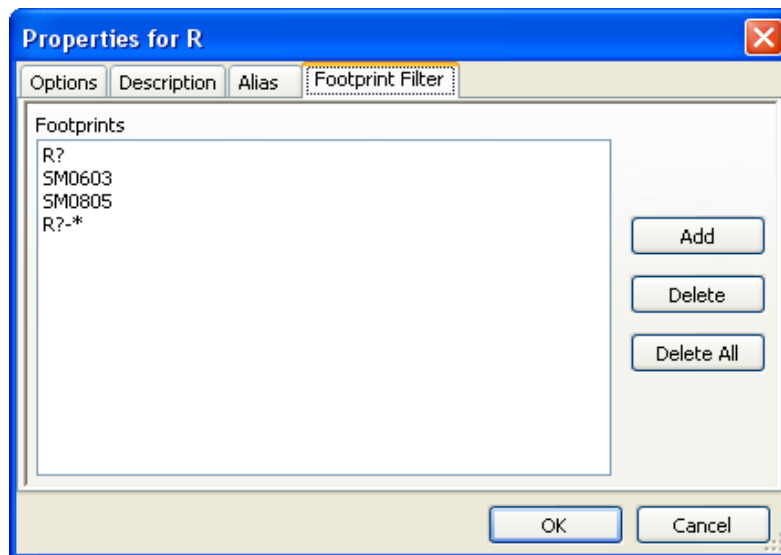In the component library editor in Eeschema, the the footprint list was set using the entries in the footprint filter tab of the component properties dialog as shown below.

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

6 - Automatic Associations

6.1 - Equivalence files

Equivalence files allow for automatic assignment of footprints to components. They list the name of the corresponding footprint according to the name (*value field*) of the component. These files typically have the .equ file extension. The are plain text files and may be edited by any plain text editor. Refer to the section "Selecting the equivalence files" for more information.

6.2 - Equivalence File Format

Equivalence files consist of one line for each component. Each line has the following structure:

'component value' 'footprint name'

Each name must be single quoted by the ' character and the component and footprint names must be separated by one or more spaces.

Example:

If the U3 component is circuit 14011 and its footprint is 14DIP300, the line is:

'14011' '14DIP300'

Any line starting with # is a comment.

Here is an example equivalence file:

```
#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'
```

6.3 - Automatically Associating Footprints to Components

Click on the automatic footprint association button on the top toolbar to process an equivalence file. All components found by their value in the selected equivalence (*.equ) file will have their footprint automatically assigned.

7 - Back-annotation File

This file can be used for the back-annotation of a schematic. This file is only used by the schematic editor Eeschema. It consists of a single line for each component containing giving the name of the footprint according to the component's reference designator.

Example:

If component U3 was assigned the footprint 14DIP300, the line generated for the component in the back-annotation file is

comp "U3" = footprint "14DIP300"

The file created has the root name of the CvPcb input net list file with extension *.stf* and is placed in the same folder as the generated net list.