



Reference manual

Copyright

This document is Copyright © 2010–2011 by its contributors as listed below. You may distribute it and/or modify it under the terms of either the GNU General Public License (*http://www.gnu.org/licenses/gpl.html*), version 3 or later, or the Creative Commons Attribution License (*http://creativecommons.org/licenses/by/3.0/*), version 3.0 or later.

All trademarks within this guide belong to their legitimate owners.

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 October 3, 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	3
2 - Cvpcb features	4
Manual or automatic association	4
Input files	4
Output files	4
3 - Invoking Cvpcb	4
4 - Cvpcb commands and usage	5
Main Cvpcb panel	5
Top toolbar	5
Cvpcb Configuration	
Cvpcb configuration screen	
Footprint library selection	
Selecting equivalence files Selecting default library path	
Selecting default library path	
Path added by users	
Paths automatically set by Cvpcb	
View the current footprint	
Additional information	
Keyboard commands Right-click menu	
Toolbar	
3D Display	
5 - Associating components with footprints	
How it works	
Assignment	
Changing an existing assignment	
Filtering the footprint list	
6 - Automatic associations	14
Equivalence files	14
File format	
Automatic component association	
7 - Back-annotation file	15

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 done and 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 one 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 can be invoked from the Kicad application pane, from the command line or directly from Eeschema. Whichever way it is chosen, Cvpcb needs a netlist file .net.

Normally a netlist file is generated starting from your schematic using Eeschema. Cvpcb will manually or automatically process it so that each component in your schematic will be associated to a PCB footprint.

From the terminal, Cvpcb can be started with the in-line command:

cvpcb <filename>

where <filename> is the name of the netlist, with or without extension .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 and usage

Main Cvpcb panel

🖾 C	vPcb (2011	-09-2	3 BZR 3145)-testi	ng F:\kicad\shar	e\demos\int	terf_u\interf_u. net 📃	
File	Preferences	Help					
ł	2 🕸		键 🗶 🍨 🛛	🐮 💣 👌			
1	BUS1	-	BUSPC :	BUS_PC		l lpin	~
2	Cl	-	47uF :	CP6		2 lpin	
3	C2	-	47pF :	C1		3 2PIN_6mm	۵
4	СЗ	-	47pF :	C1		4 3M-N7E50	
5	C4	-	47uF :	CP6		5 3PIN_6mm	
6	C5	-	47uF :	CP6		6 8DIPCMS	
7	C6	-	47uF :	CP6		7 20TEX-ELL300	
8	D1	-	LED :	LEDV		8 20TEX300	
9	D2	-	LED :	LEDV		9 24tex300	
10	Gl	-	L0G0 :	LOGO		10 24TEXT-E11300	
11	JP1	-	CONN_8X2 :	pin_array_8x2		11 28TEX-E11600	
12	Pl	-	DB25FEMELLE :	DB25FC		12 28tex600	
13	Rl	-	100K :	R3		13 40tex-E11600	
14	R2		1K :	R3		14 40tex600	
15	R3	-	10K :	R3		15 80188	
16	R4	-	330 :	R3		16 ADSP2100	
17	R5	-	330 :	R3		17 AFF_2x7SEG-DIGIT_10m	m
18	RR1	-	9x1K :	r_pack9		18 AK300-2	
19	Ul	-	74LS245 :	DIP-20300		19 BGA48	
20	U2	-	74LS688 :	DIP-20300		20 BGA64-0.8mm	
21	U3	-	74LS541 :	DIP-20_300		21 BGA90-0.8	
22	U5	-	628128 :	DIP-32_600		22 BGA121_1mm	
23	U 8	-	EP600 :	DIP-24_300		23 BGA144_1mm	
24	U9	-	4003APG120 :	PGA120		24 BGA256	
25	Xl	-	8MHz :	HC-18UH		25 BGA352	
						26 BGA400_1mm	
						27 DOMAN 1mm	~
						<	>
Compo	onents: 25 (fre	ee: 0)			Footprint	s (All): 456	

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

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

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

Top toolbar



The top toolbar allows for the execution of the following commands:



Select the netlist file to be processed.

	Save the assignment file .cmp and the updated Netlist file .net
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~	Invoke the Cvpcb configuration menu.
	Display the footprint of the component selected in the footprint window.
縺	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.
<b>F</b>	Delete all assignments.
ť	Generate footprint assignment back-annotation file.
	Open the footprint documentation pdf file using the default pdf viewer.
<b>:</b>	Enable or disable the footprint filtering to display the list of available 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 it will display the following screen:

Project file: F:\kicad\share\demos\interf_u\interf_u.pro	
Footprint library files	
connect dip_sockets	Add
discret	Insert
pin_array divers	Remove
Liberne	
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

#### Footprint library selection

connect dip_sockets sockets discret pin_array divers		Add Insert Remove
---------------------------------------------------------------------	--	-------------------------

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

- Footprint alias files	
devcms	Add
	Insert
	Remove

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.

Cuser 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	

#### Path added by users

-L	Iser defined search paths	
	F:\kicad\share\modules	Add
		Insert
		Remove

#### Paths automatically set by Cvpcb

They partially depend on 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

#### View the current footprint

The View command allows you 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.

You can also display the 3D view, if it has been created and assigned to the footprint.

🖀 Module: DIP-1	4300_ELL	
🛃 ବ୍ ବ୍		
		· · · ·
In mm ↓		
T O	DIP-14_300_ELL	
		~
U*** DIP-14_300_ELL	Dernier Changement Couche Pads Stat Orient Module Forme 3D Jan 25, 1970 Dessus 14 0,0 DIP-14_300_ELL dil/dil_14.wrl	Doc: 14 pins D KeyW: DIL
Lib: F:\kica Z 15	X 0,0000 Y 0,0000 dx 0,0000 dy 0,0000 Pouces	

### Additional information

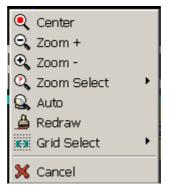
The coordinates of the cursor are displayed at the bottom of the screen: Absolute coordinates (X nnnn Y nnnn) and relative coordinates (dx nnnn dy nnnn)

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

#### Keyboard commands

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

#### Right-click menu



Displayed by right-clicking the mouse:

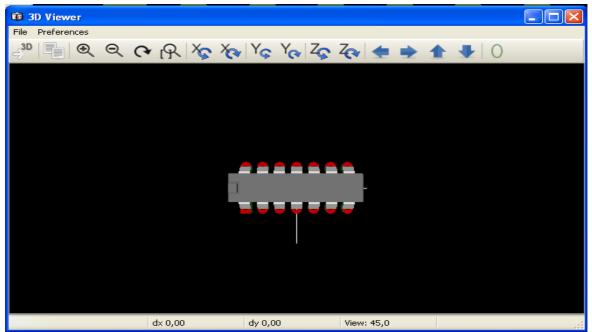
Zoom Selection (Select Zoom)	Direct selection of the display zoom .
Grid Selection (Grid Select)	Direct selection of the grid.

#### Toolbar



	Display options
	Zoom levels
30	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:

🚟 CvPcb (2011-09-23 BZR 3145)-testing F:\kicad\share\demos\interf_u\interf_u.net 📃 🗖 🔀							
File	Preferences	Help					
ł¢	2	<u> </u>	親 🛨 单	🐮 💣 👌			
1	BUS1	-	BUSPC :	BUS_PC		l lpin	~
2	C1	-	47uF :	CP6		2 lpin	
3	C2	-	47pF :	C1		3 2PIN_6mm	
4	СЗ	-	47pF :	C1		4 3M-N7E50	
5	C4	-	47uF :	CP6		5 3PIN_6mm	
6	C5	-	47uF :	CP6		6 SDIPCMS	
7	Ce		47uF :			7 20TEX-ELL300	
8	Dl	-	LED :	LEDV		8 20TEX300	
9	D2	-	LED :	LEDV		9 24tex300	
10	G1	-	L0G0 :	LOGO		10 24TEXT-E11300	
11	JP1	-	CONN_8X2 :	pin_array_8x2		11 28TEX-E11600	
12	Pl	-	DB25FEMELLE :	DB25FC		12 28tex600	
13	Rl	-	100K :	R3		13 40tex-E11600	
14	R2	-	1K :	R3		14 40tex600	
15	R3	-	10K :	R3		15 80188	
16	R4	-	330 :	R3		16 ADSP2100	
17	R5	-	330 :	R3		17 AFF_2x7SEG-DIGIT_10m	m
18	RR1	-	9x1K :	r_pack9		18 AK300-2	
19	U1	-	74LS245 :	DIP-20300		19 BGA48	
20	U2	-	74LS688 :	DIP-20300		20 BGA64-0.8mm	
21	US	-	74LS541 :	DIP-20300		21 BGA90-0.8	
22	U5	-	628128 :	DIP-32600		22 BGA121_1mm	
23	U8	-	EP600 :	DIP-24300		23 BGA144_1mm	
24	U9	-	4003APG120 :	PGA120		24 BGA256	
25	X1	-	SMHz :	HC-18UH		25 BGA352	
						26 BGA400_1mm	~
						27 DC3404 1mm	
							<u> </u>
Components: 25 (free: 0) Footprints (All): 456							

#### With filtering:

🗠 CvPcb (2011-09-23 BZR 3145)-testing F:\kicad\share\demos\interf_u\interf_u.net								
File Preferences Help								
15 🖄 🎯 💓 🔙 🍝 🔶 🚼 💣 躍								
1 BUS1 - BUSPC : BUS_PC	1 R1							
2 C1 - 47uF : CP6	2 R3							
3 C2 - 47pF : C1	3 R3-5							
4 C3 - 47pF : C1	4 R3-LARGE_PADS							
5 C4 - 47uF : CP6	5 R4							
6 C5 - 47uF : CP6	6 R4-5							
7 C6 - 47uF : CP6	7 R4-LARGE_PADS							
8 D1 - LED : LEDV	8 R5							
9 D2 - LED : LEDV	9 R6							
10 G1 - LOGO : LOGO	10 R7							
<pre>11 JP1 - CONN_8X2 : pin_array_8x2</pre>	11 SM0603							
12 P1 - DB25FEMELLE : DB25FC	12 SM0805							
13 R1 - 100K : R3	13 SM1206							
14 R2 - 1K : R3								
15 R3 - 10K : R3								
16 R4 - 330 : R3								
17 R5 - 330 : R3								
18 RR1 - 9x1K : r_pack9								
19 Ul - 74LS245 : DIP-20_300								
20 U2 - 74LS688 : DIP-20_300								
21 U3 - 74LS541 : DIP-20_300								
22 U5 - 628128 : DIP-32_600								
23 U8 - EP600 : DIP-24_300								
24 U9 - 4003APG120 : PGA120								
25 X1 - 8MHz : HC-18UH								
Components: 25 (free: 0) Footprints (filtered): 13								

Under Eeschema, the allowed footprint list was:

Propriétés pour R 🛛 🔀								
Options Doc Alias Champs Filtrage Modules								
Modules R? SM0803 SM0805 Ajouter Supprimer Tout Supprimer								
Annuler OK								

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 the automatic assignment. They provide 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.

### **File 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' 'S014E' '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-LOFP100' 'VOFP100' #regulators 'LP2985LV' 'SOT23-5'

#### Automatic component association

The automatic association process is enabled by clicking on the icon

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

## 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 one 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.