



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 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	3
2 - Cvpcb features	4
Manual or automatic association	4
Input files	4
Output files	4
3 - Invoking Cvpcb	4
4 - Cvpcb Commands	5
Main screen	5
Main top toolbar	5
Cvpcb Configuration	6
Cvpcb configuration screen	6
Footprint library selection	7
Selecting equivalence mes	o o
Selecting default library path	o 8
Path added by users	8
Paths automatically set by Cvpcb	9
Viewing the current footprint	9
Additional information	
Keyboard commands Pight-click monu	10
Toolbar.	
3D Display	
5 - Associating components with footprints	
How it works	
Assignment	
Changing an existing assignment	
Filtering the footprint list	12
6 - Automatic associations	
Equivalence files	
=q====================================	
Automatic component association	
7 - Back-appotation 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 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

🖀 C	vPcb (2011	-09-23 BZR	3145)-testi	ng F:\kicad\	share\demo	os\inte	erf_u'	\interf_u.net		\mathbf{X}
File	Preferences	Help								
ł	١	الله 🍋	* *	8						
1	BUS1	-	BUSPC :	BUS_PC			1	lpin		~
2	C1	-	47uF :	CP6			2	lpin		
3	C2	-	47pF :	C1			3	2PIN_6mm		
4	C3	-	47pF :	C1			4	3M-N7E50		
5	C4	-	47uF :	CP6			5	3PIN_6mm		
6	C5	-	47uF :	СРб			6	SDIPCMS		
7	C6	-	47uF :	CP6			7	20TEX-ELL300		
8	Dl	-	LED :	LEDV			8	20 TE X300		
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	- DB2	SFEMELLE :	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_	10mm	
18	RR1	-	9x1K :	r_pack9			18	AK300-2		
19	U1	-	74LS245 :	DIP-2030	0		19	BGA48		
20	U2	-	74LS688 :	DIP-2030	0		20	BGA64-0.8mm		
21	U3	-	74LS541 :	DIP-2030	0		21	BGA90-0.8		
22	U5	-	628128 :	DIP-32_60	0		22	BGA121_1mm		
23	U8	-	EP600 :	DIP-2430	0		23	BGA144_1mm		
24	U9	- 40	03APG120 :	PGA120			24	BGAZ56		
25	X1	-	SMHz :	HC-18UH			25	BGA352		
							26	BGA400_1mm		~
							<			
Compo	onents: 25 (fro	ee: 0)			Foo	otprints	(All): 4	56		.;;

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.
繿	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.
ſ	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:

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

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.

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	

Path added by users

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

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.

🚟 Module:	DIP-14	300_ELL						
۹ 🛃	90	¥ (Q. <mark>3</mark> [
							<u> </u>	
		44			R (6	8	
		٦	U**	*				
T			DI	P-1	4_3	00_E		
			R	1	9 6			
()		Derpier Chape	ement Couch	e Dada	Stat Orient	Module	Forme 3D	Doc: 14 pipe D
DIP-14_300	_ELL	Jan 25, 1970	Dessu	is 14	0,0	DIP-14300_ELL	dil/dil_14.wrl	KeyW: DIL
Lib: F:\kica	Z 15	X 0,000	0 Y 0,0000	dx 0,00	000 dy 0,0000	Pouces	5	

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	coom 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:

🚰 C	/Pcb (2011	-09-23 BZR 3145)-testing F:\kicad\s	.hare\demos\interf_u\interf_u.net 📃 🗖 🔀					
File	Preferences	Help						
ł	٢	🔛 💓 🗢 🏓 🚼 💣 🛛						
1	BUS1	- BUSPC : BUS_PC	l lpin 🔨					
2	C1	- 47uF : CP6	2 lpin					
3	C2	- 47pF : Cl	3 2PIN_6mm 🥮					
4	C3	- 47pF : Cl	4 3M-N7E50					
5	C4	- 47uF : CP6	5 3PIN_6mm					
6	C5	- 47uF : CP6	6 SDIPCMS					
7	C6	- 47uF : CP6	7 20TEX-ELL300					
8	Dl	- LED : LEDV	8 20TEX300					
9	D2	- LED : LEDV	9 24tex300					
10	G1	- LOGO : LOGO	10 24TEXT-E11300					
11	JP1	 CONN_8X2 : pin_array_8 	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_10mm					
18	RR1	- 9x1K : r_pack9	18 AK300-2					
19	01	- 74LS245 : DIP-20_300	19 BGA48					
20	U2	- 74LS688 : DIP-20_300	20 BGA64-0.8mm					
21	U3	- 74LS541 : DIP-20_300	21 BGA90-0.8					
22	05	- 628128 : D1P-32_600	22 BGA121_1mm					
23	08	- KP600 : D1P-24_300	23 BGA144_1mm					
24	09	- 4003APG120 : PGA120	24 BGA256					
25	XI	- SMHZ : HC-180H	25 BGA352					
			25 BGA400_1mm					
Components: 25 (free: 0) Footprints (All): 456								

With filtering:

🖾 C	vPcb (2011	1-09-23 BZR	R 3145)-testi	ing F:\kicad\share\d	emos\inte	erf_u	\interf_u.net		
File	Preferences	Help							
ł	2	3	* *	🗜 矿 👌 🛱					
1	BUS1	-	BUSPC :	BUS_PC		1	Rl		
2	C1	-	47uF :	CP6		2	R3		
3	C2	-	47pF :	Cl		3	R3-5		
4	СЗ	-	47pF :	Cl		4	R3-LARGE_PADS		
5	C4	-	47uF :	CP6		5	R4		
6	C5	-	47uF :	CP6		6	R4-5		
7	Ce	-	47uF :	CP6		7	R4-LARGE_PADS		
8	Dl	-	LED :	LEDV		8	R5		
9	D2	-	LED :	LEDV		9	R6		
10	G1	-	L0G0 :	LOGO		10	R7		
11	JP1	-	CONN_8X2 :	pin_array_8x2		11	SM0603		
12	Pl	- DB2	25FEMELLE :	DB25FC		12	SM0805		
13	Rl	-	100K :	R3		13	SM1206		
14	R2	-	1K :	R3					
15	R3	-	10K :	R3					
16	R4	-	330 :	R3					
17	RS DD1	-	330 :	R3					
18	RRI	-	9XIK :	r_pack9					
20	112	2	7465245 .	DIP-20300					
20	112	_	74136666 .	DIP-20300					
22	115	_	628128 -	DTP-32 600					
23	118	-	RP600 -	DTP-24 300					
24	119	- 40	03APG120 -	PGA120					
25	XI	-	8MHz :	HC-18UH					
Compo	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 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' '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-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.

Back-annotation file 7 -

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.