

GerbView

October 19, 2016

Contents

1 Introduction to GerbView **2**

2 Main Screen **2**

3 Top toolbar **3**

4 Left toolbar **4**

5 Commands in menu bar **5**

5.1 File menu 5

5.1.1 Export to Pcbnew 5

5.2 Preferences menu 5

5.3 Miscellaneous menu 6

6 Layer Manager **7**

6.1 Modes to show Gerber layers 8

6.2 Effect of layer selection for drawings 9

7 Print layers **10**

7.1 Print dialog access 10

7.2 Move block command 10

Reference manual

Copyright

This document is Copyright © 2010-2015 by it's 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

The KiCad Team.

Feedback

Please direct any bug reports, suggestions or new versions to here:

- About KiCad document: https://github.com/KiCad/kicad-doc/issues
- About KiCad software: https://bugs.launchpad.net/kicad
- About KiCad software i18n: https://github.com/KiCad/kicad-i18n/issues

Publication date and software version

Published on February 14, 2015.

1 Introduction to GerbView

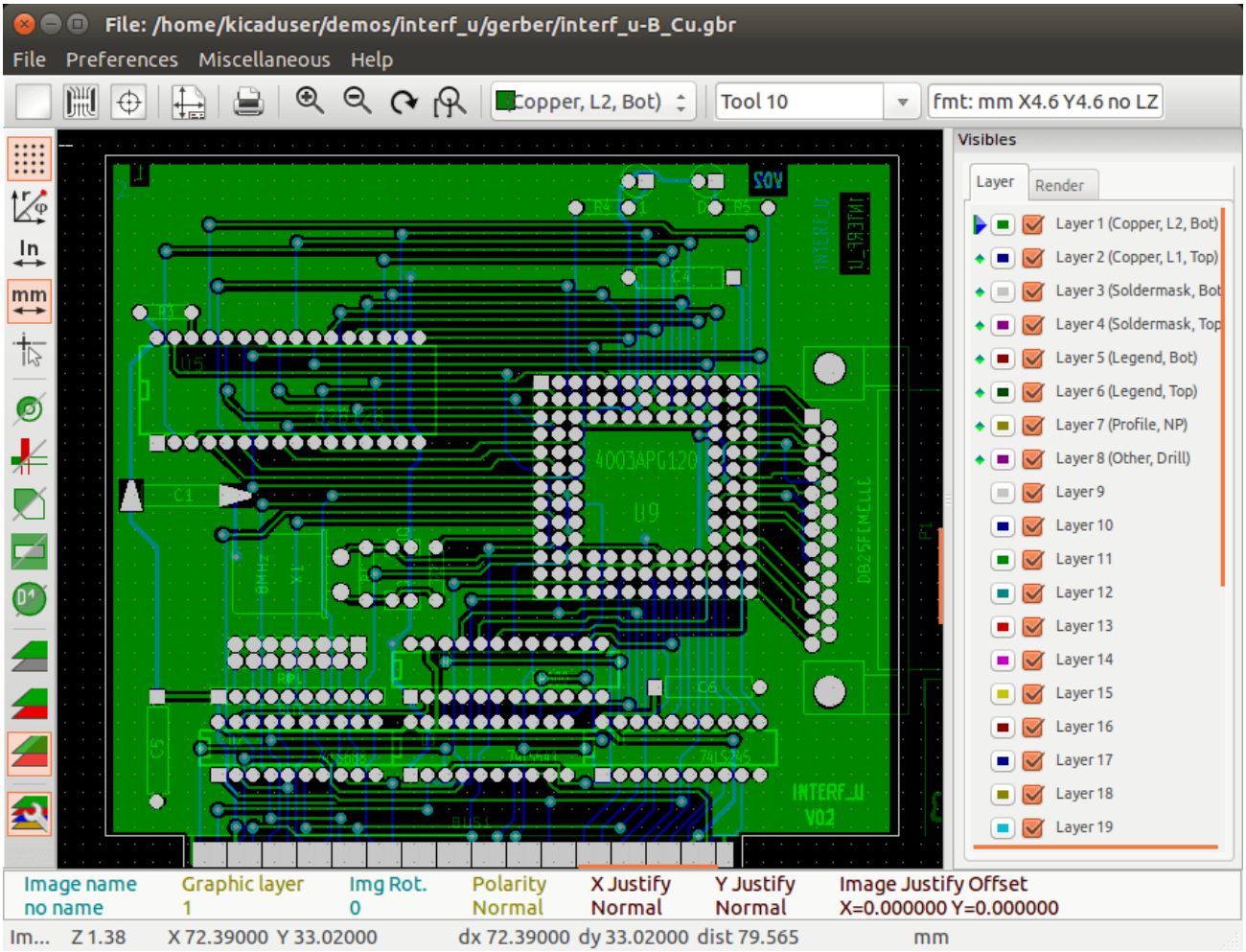
GerbView is a Gerber file viewer (RS 274 X format), and is also able to display drill files from Pcbnew (in Excellon format).

It accepts up to 32 files (Gerber and/or Drill files)

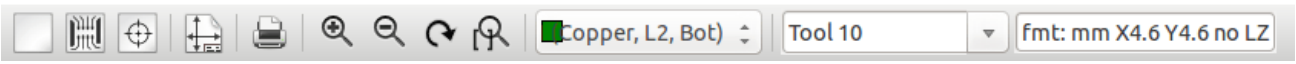
Files can be displayed using a transparency mode or stacked mode.

For more information about the Gerber file format please have a read at the specification in The Gerber File Format Specification - Ucamco.

2 Main Screen



3 Top toolbar



	Clear all layers
	Load Gerber files
	Load drill files (Excellon format from Pcbnew)
	Set page size for printing and show/hide page limits
	Open print dialog
	Zoom in and out
	Refresh screen
	Zoom fit in page
Copper, L2, Bot)	Layer selection
Tool 10	D Code selection (hight light items that use this dcode)
fmt: mm X4.6 Y4.6 no LZ	Info about Gerber file options loaded in the current layer

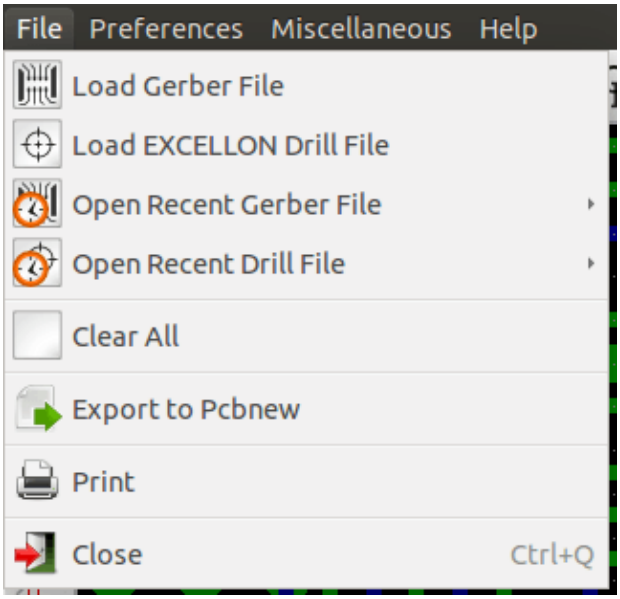
4 Left toolbar



	Grid on / off
	Display polar coordinates on / off
	Units selection to display coordinates
	On grid cursor shape selection
	Display mode selection (solid or outlines) for flashed items
	Display mode selection (solid or outlines) for lines
	Display mode selection (solid or outlines) for polygons
	Show negative objects in ghost color
	Show / hide D Codes values (for items using a dcode)
	Mode used by Gerbview to show layers.

5 Commands in menu bar

5.1 File menu



It is possible to load gerber and drill files into Gerbview. There is also an auxiliary option to export gerbers to pcbnew. Previously (a long time ago) it was also possible to load so called Dcodes, but those are now obsolete and is therefore not possible anymore.

5.1.1 Export to Pcbnew

GerbView has a limited capability to export Gerber files to Pcbnew.

The final result depends on what features of RS 274 X format are used in Gerber Files.

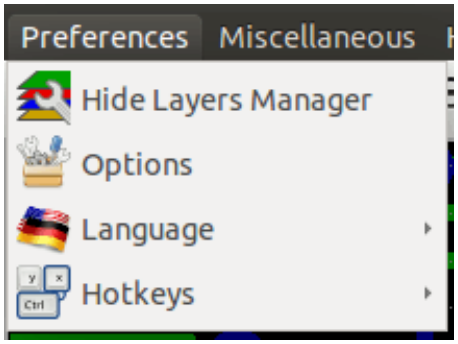
RS 274 X format has raster oriented features that cannot be converted (mainly all features relative to negative objects).

Flashed items are converted to vias.

Lines items are converted to track segments (or graphic lines for non copper layers)

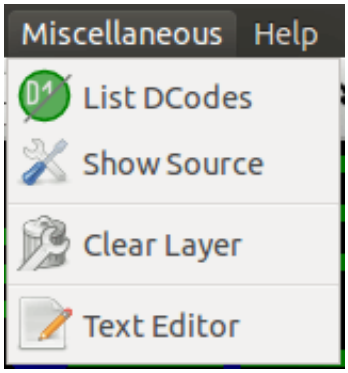
So the usability of the converted file is very dependent upon the way each Gerber file was built by the original Pcb tool.

5.2 Preferences menu



Gives access to the hot keys editor, and some options to display items.

5.3 Miscellaneous menu

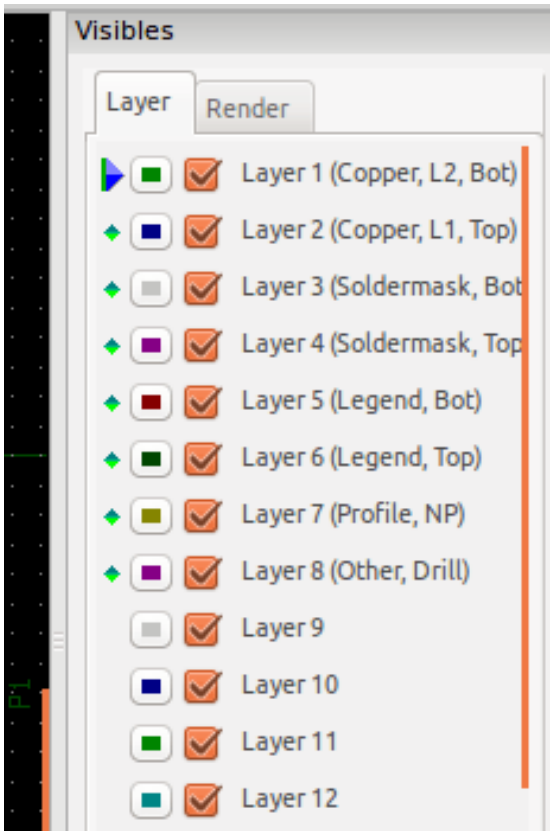


- List Dcodes shows the Dcodes in use and some of Dcode parameters.
- Show Source displays the Gerber file contents of the active layer in a text editor.
- Clear Layer erases the contents of the active layer.

6 Layer Manager

The layer manager has 2 purposes:

- Select the active layer
- Show/hide layers



The active layer is drawn after the other layers.

When loading a new file, the active layer is used (the new data replace the previous data)

Note:

- Mouse left click on a line: select the active layer
- Mouse right click on the layer manager: show/hide all layers
- Mouse middle click on a icon: select the layer color.

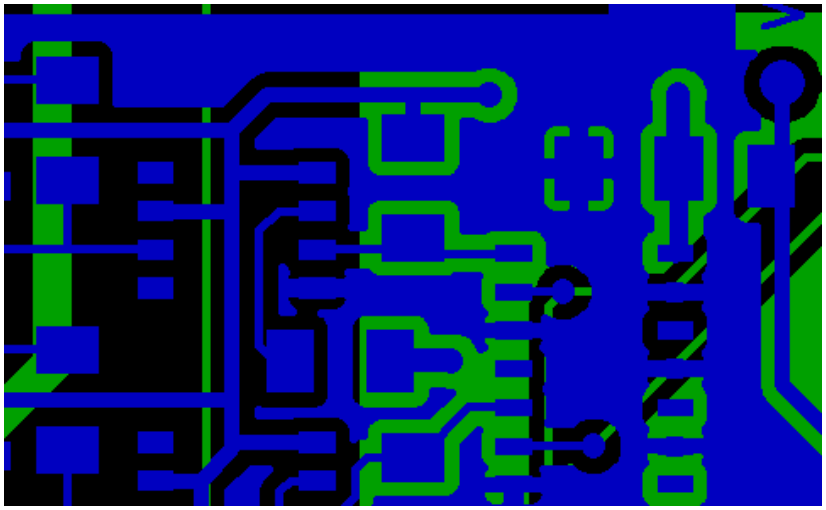
6.1 Modes to show Gerber layers

- Raw mode 

Each gerber file and each item in files are drawn in the order files are loaded.

However the **active layer** is draw last.

When Gerber files have negative items (drawn in black) artefacts are visible on already drawn layers

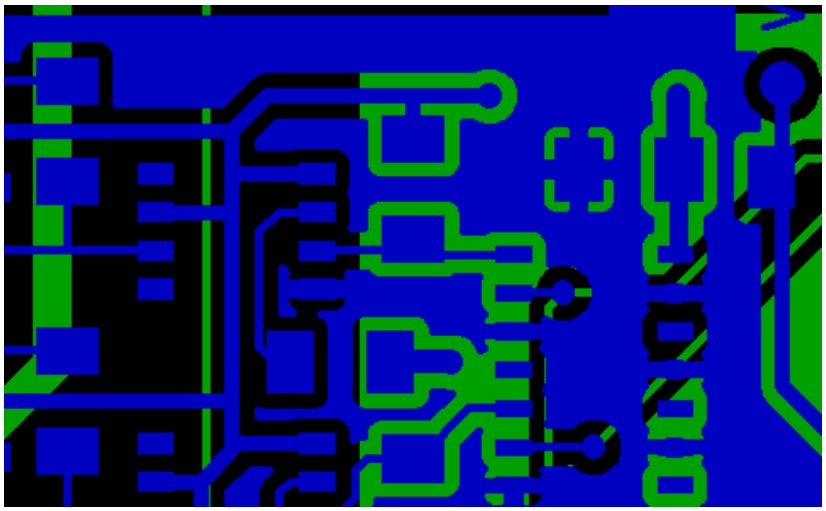


- Stacked mode 

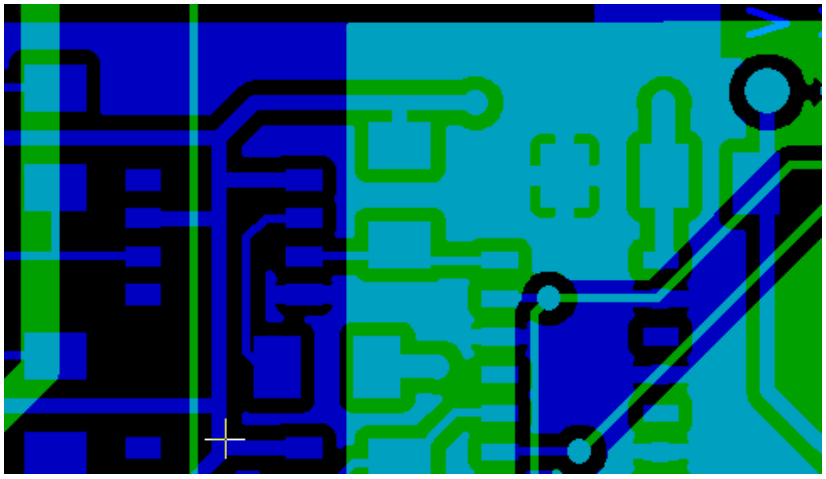
Each gerber file is drawn in the order files are loaded.

The **active layer** is draw last.

When Gerber files have negative items (drawn in black) there are no artefacts on already drawn layers, because this mode draws each file in a local buffer before it is shown on screen. Negative items do not create artefacts.



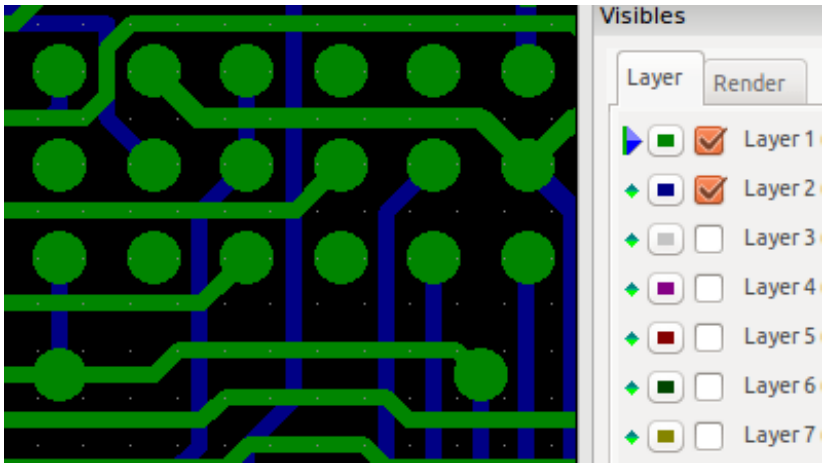
- Transparency mode 



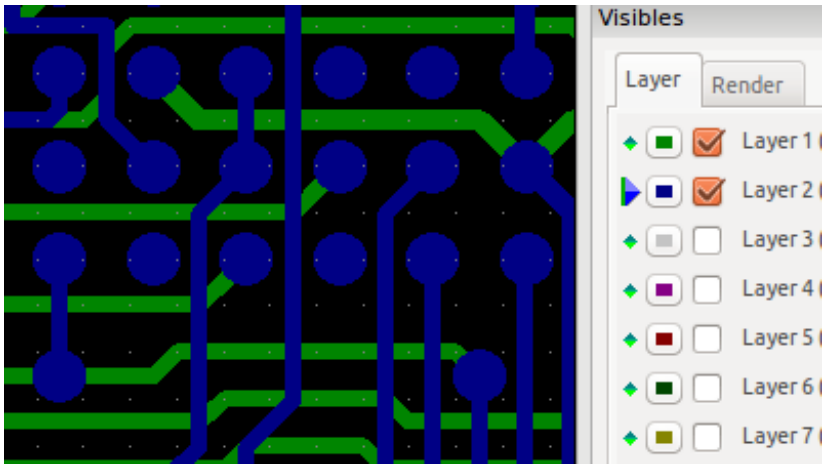
6.2 Effect of layer selection for drawings

This effect is visible only in raw or stacked mode.

The layer 1 (green layer) is drawn after the layer 2



The layer 2 (blue layer) is drawn after the layer 1



7 Print layers

7.1 Print dialog access

To print layers, use the  tool, or the main menu (files)



Caution

But be sure items are inside the printable area (select by  a suitable page format).
Do not forget photoplotters can use a large plottable area, much bigger than the page sizes used by printers)
Moving (by block move command) the entire layers is often needed.

7.2 Move block command

You can move items by selecting them (drag the mouse with left button down) and then moving the selected area on screen.

Click the left button to finally place the area you are moving.