Gerber Viewer

The KiCad Team

REVISION HISTORY			
NUMBER	DATE	DESCRIPTION	NAME

Contents

1 Introducción a GerbView **2**

2 Interface **2**

2.1 Main window 2

2.2 Barra de herramientas superior 3

2.3 Barra de herramientas izquierda 4

2.4 Layers Manager 5

3 Comandos de la barra de menús **6**

3.1 Menú archivo 6

3.2 Tools menu 6

4 Printing **7**

Manual de referencia

Copyright

This document is Copyright © 2010-2021 by its contributors as listed below. You may distribute it and/or modify it under the terms of either the GNU General Public License (https://www.gnu.org/licenses/gpl.html), version 3 or later, or the Creative Commons Attribution License (https://creativecommons.org/licenses/by/3.0/), version 3.0 or later.

Todas las marcas mencionadas en esta guía pertenecen a sus legítimos propietarios.

Contributors

El Equipo de KiCad.

Traducción

Antonio Morales <antonio1010.mr@gmail.com>, 2015-2016

Feedback

The KiCad project welcomes feedback, bug reports, and suggestions related to the software or its documentation. For more information on how to sumbit feedback or report an issue, please see the instructions at https://www.kicad.org/help/report-an-issue/

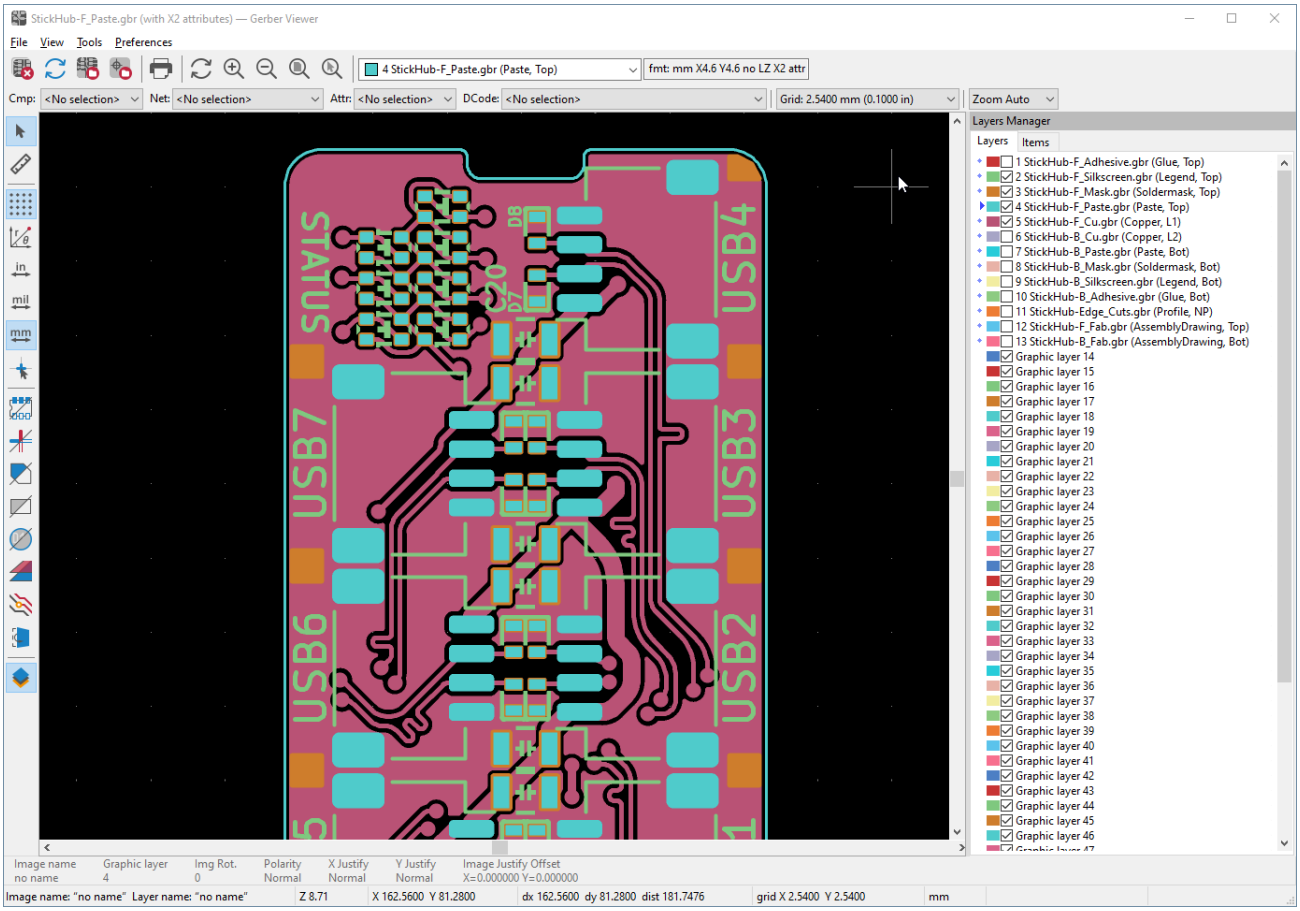
1 Introducción a GerbView

GerbView is a Gerber file (RS-274X format) and Excellon drill file viewer. Up to 32 files can be displayed at once.

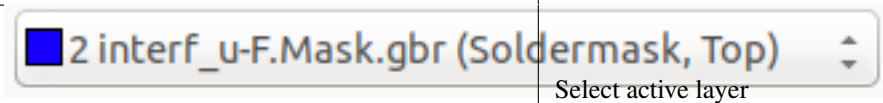
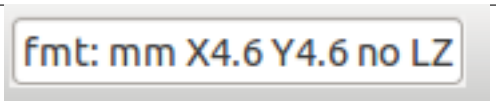
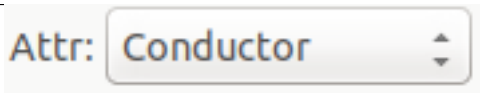
For more information about the Gerber file format please read the Gerber File Format Specification. Details about drill file format can be found at the Excellon format description.

2 Interface

2.1 Main window



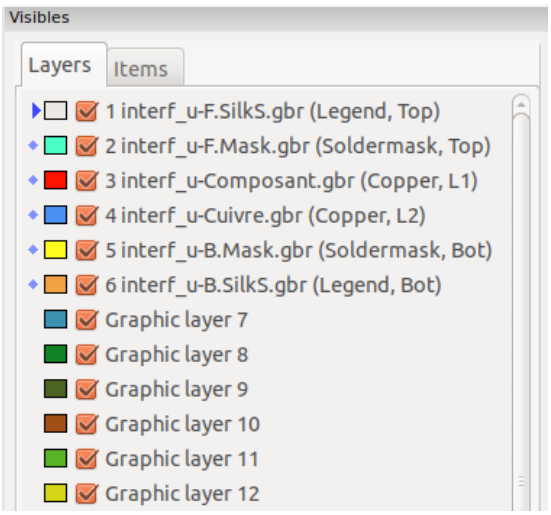
2.2 Barra de herramientas superior

	Clear all layers
	Load Gerber files
	Load Excellon drill files
	Set page size
	Print
	Redraw view
	Zoom in or out
	Zoom to fit page
	Zoom to selection
	Select active layer
	Display info about active layer
	Highlight items belonging to selected component (Gerber X2)
	Highlight items belonging to selected net (Gerber X2)
	Highlight items with the selected attribute (Gerber X2)
	Highlight items of selected D Code on the active layer

2.3 Barra de herramientas izquierda

	Select items
	Measure between two points
	Toggle grid visibility
	Toggle polar coordinates display
	Select inch, mils, or millimeter units
	Toggle full-screen cursor
	Display flashed items in sketch (outline) mode
	Display lines in sketch (outline) mode
	Display polygons in sketch (outline) mode
	Show negative objects in ghost color
	Show/hide D Codes
	Display layers in diff (compare) mode
	Toggle inactive layers between normal and dimmed display
	Show/hide layer manager
	Show Gerbers as mirror image

2.4 Layers Manager



The Layers Manager controls and displays visibility of all layers. An arrow indicates the active layer, and each layer can be shown or hidden with the checkboxes.

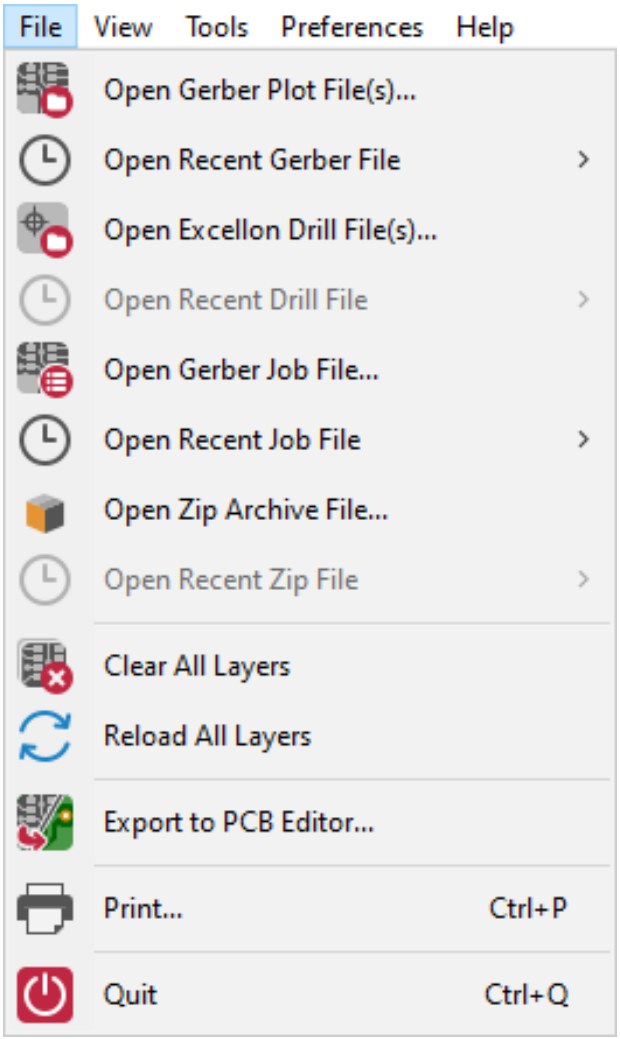
Mouse button assignments:

- Left click: select the active layer
- Right click: show/hide/sort layers options
- Middle click or double click (on color swatch): select the layer color

The Layers tab allows you to control the visibility and color of all loaded Gerber and drill layers. The Items tab allows you to control the color and display of the grid, D Codes, and negative objects.

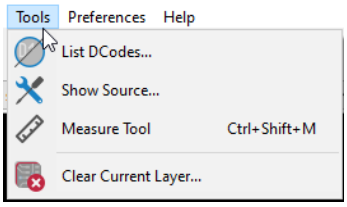
3 Comandos de la barra de menús

3.1 Menú archivo



- **Export to PCB Editor** is a limited capability to export Gerber files into a KiCad PCB. The final result depends on what features of the RS-274X format are used in the original Gerber files: rasterized items cannot be converted (typically negative objects), flashed items are converted to vias, lines are converted to track segments (or graphic lines for non-copper layers).

3.2 Tools menu



- **List DCodes** shows the D Code information for all layers.
- **Show Source** displays the Gerber file contents of the active layer in a text editor.
- **Measure Tool** allows measuring the distance between two points.
- **Clear Current Layer** erases the contents of the active layer.

4 Printing

To print layers, use the  icon or the **File** → **Print** menu.



Caution

Be sure items are inside the printable area. Use  to select a suitable page format.
Note that many photoplotters support a large plottable area, much bigger than the page sizes used by most printers. Moving the entire layer set may be required.