

GerbView

Table of Contents

Introduzione a GerbView	2
Interfaccia	2
Finestra principale	2
Barra superiore	3
Barra strumenti sinistra	4
Gestione strati	5
Comandi nella barra del menu	5
Menu file	6
Tools menu	6
Stampa	7

Manuale di riferimento

Copyright

This document is Copyright © 2010-2021 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 (<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.

Tutti i marchi registrati all'interno di questa guida appartengono ai loro legittimi proprietari.

Collaboratori

Il Team di KiCad.

Traduzione

Marco Ciampa <ciampix@posteo.net>, 2014-2018.

Feedback

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

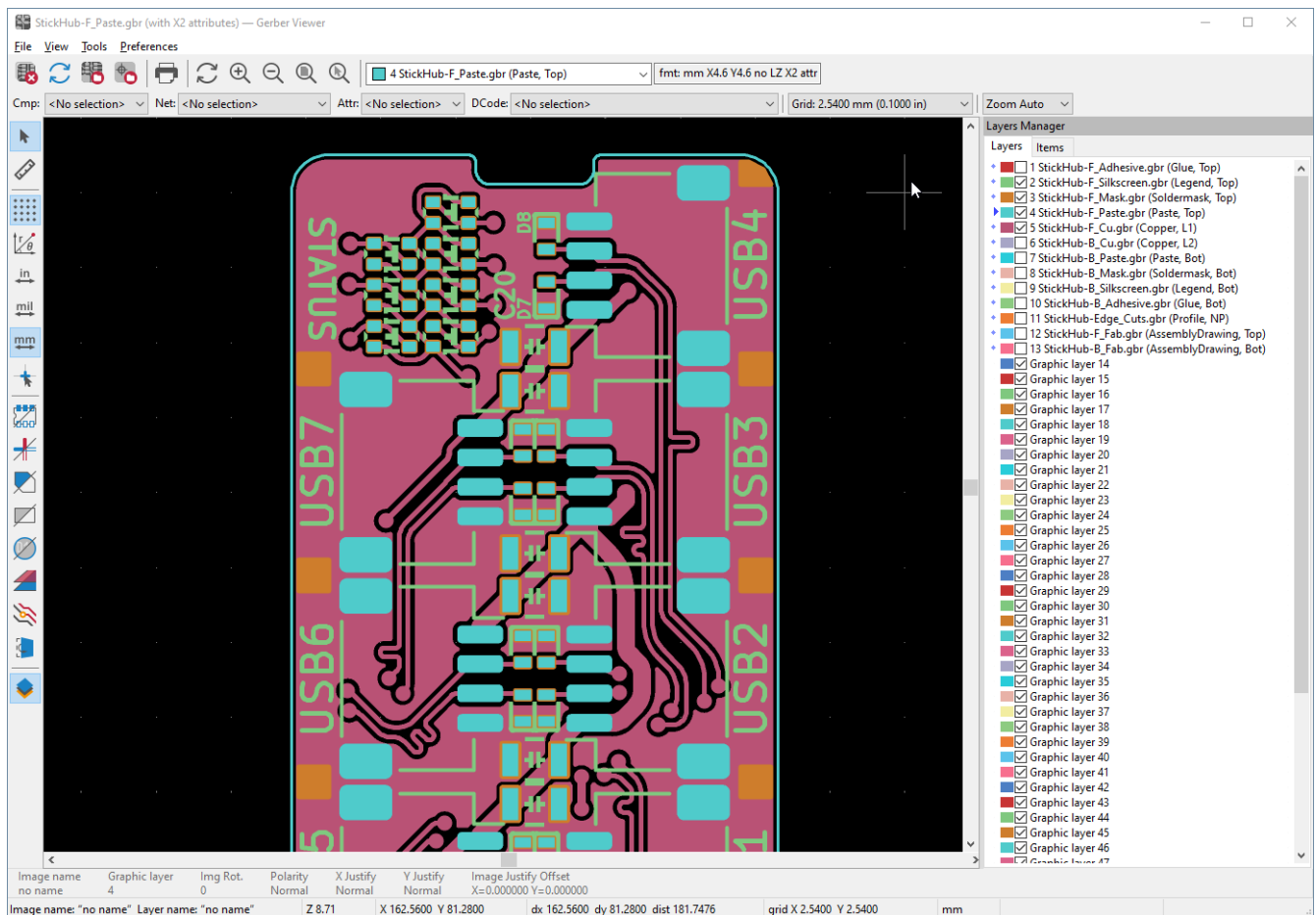
Introduzione a GerbView

GerbView è un visualizzatore di file in formato Gerber (formato RS 274 X), ed è anche in grado di visualizzare i file forature generati da Pcbnew (in formato Excellon).










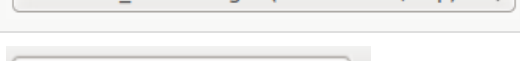
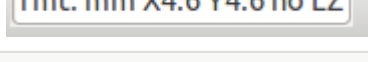
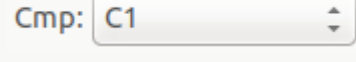
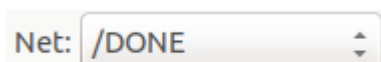
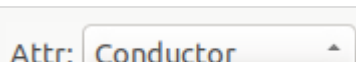
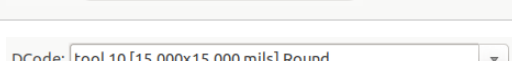
Per ulteriori informazioni sul formato file Gerber consultare [Le specifiche del formato file Gerber - Ucamco](#).

Interfaccia
















Finestra principale



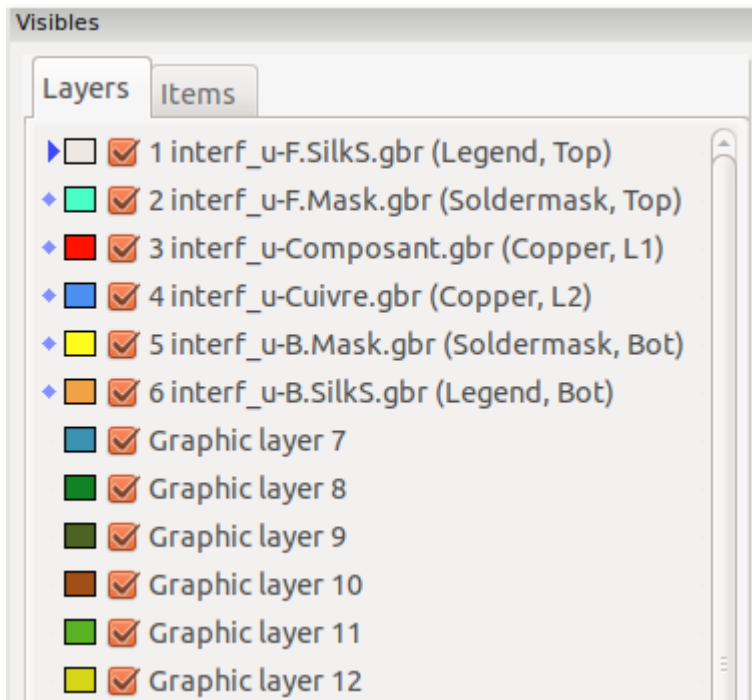
Barra superiore

	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

Barra strumenti sinistra

	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

Gestione strati



La gestione strati controlla la visibilità di tutti gli strati. Una freccia indica lo strato attivo, ed ogni strato può essere mostrato o nascosto tramite le caselle a spunta.

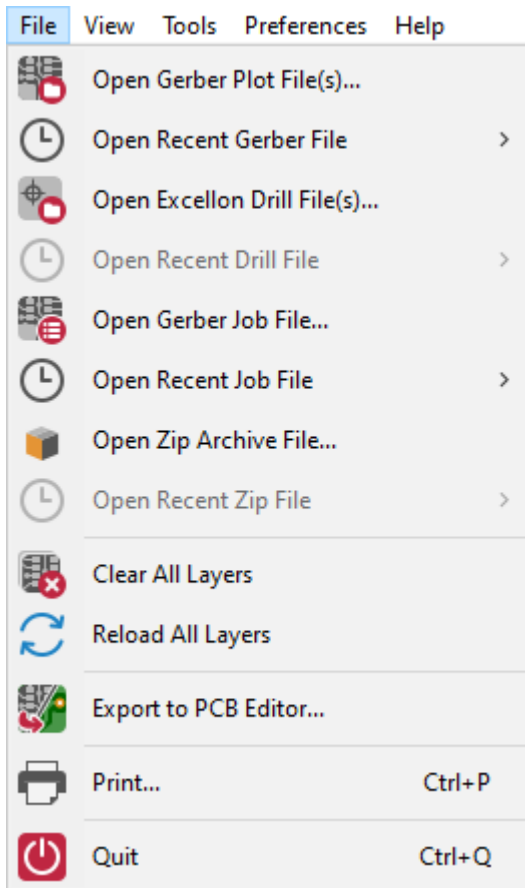
Assegnazione pulsanti del mouse:

- Clic sinistro del mouse: seleziona lo strato attivo
- Clic destro: opzioni mostra/nascondi/ordina strati
- Clic centrale del mouse o doppio clic (su un campione di colore): seleziona il colore dello strato

La scheda strati permette di controllare la visibilità ed i colori di tutti gli strati Gerber e forature caricati. La scheda elementi permette di controllare il colore e la visibilità della griglia, Codici D, e oggetti negativi.

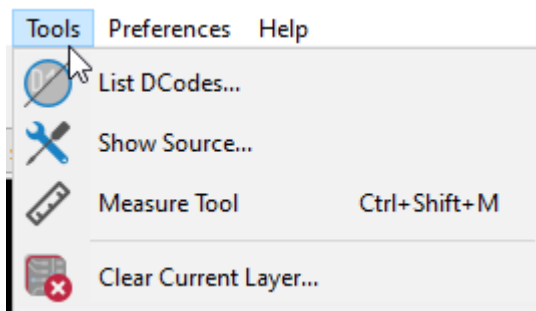
Comandi nella barra del menu

Menu file



- **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).

Tools menu



- **Lista Dcode** mostra le informazioni sui Dcode per tutti gli strati.
- **Show Source** displays the Gerber file contents of the active layer in a text editor.
- **Measure Tool** allows measuring the distance between two points.
- **Pulisci strato corrente** cancella i contenuti dello strato attivo.

Stampa

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

Be sure items are inside the printable area. Use  to select a suitable page format.

CAUTION

Si noti che diversi photoplotter supportano un'area di disegno molto ampia, molto più ampia delle dimensioni della pagina usate dalla maggior parte delle stampanti. Può rendersi necessario spostare l'intero insieme di strati.