Gerber Viewer

The KiCad Team

REVISION HISTORY			
NUMBER	DATE	DESCRIPTION	NAME

Contents

1 GerbView の紹介 2

2 インタフェース 2

2.1 メイン・ウィンドウ 2

2.2 上部ツールバー 3

2.3 左ツールバー 4

2.4 レイヤー・マネージャー . . . 5

3 メニュー・バーのコマンド 5

3.1 ファイル・メニュー 5

3.2 Tools menu 5

4 印刷 6

リファレンス・マニュアル

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.

このガイドの中のすべての商&#x

Contributors

The KiCad Team.

翻訳

murakou <murakou AT jp3cyc.jp>, 2018. starfort <starfort AT nifty.com>, 2017-2018. kinichiro <kinichiro.inoguchi AT gmail.com>, 2015. Norio Suzuki <nosuzuki AT postcard.st>, 2015. yoneken <yoneken AT kicad.jp>, 2011-2015.

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/

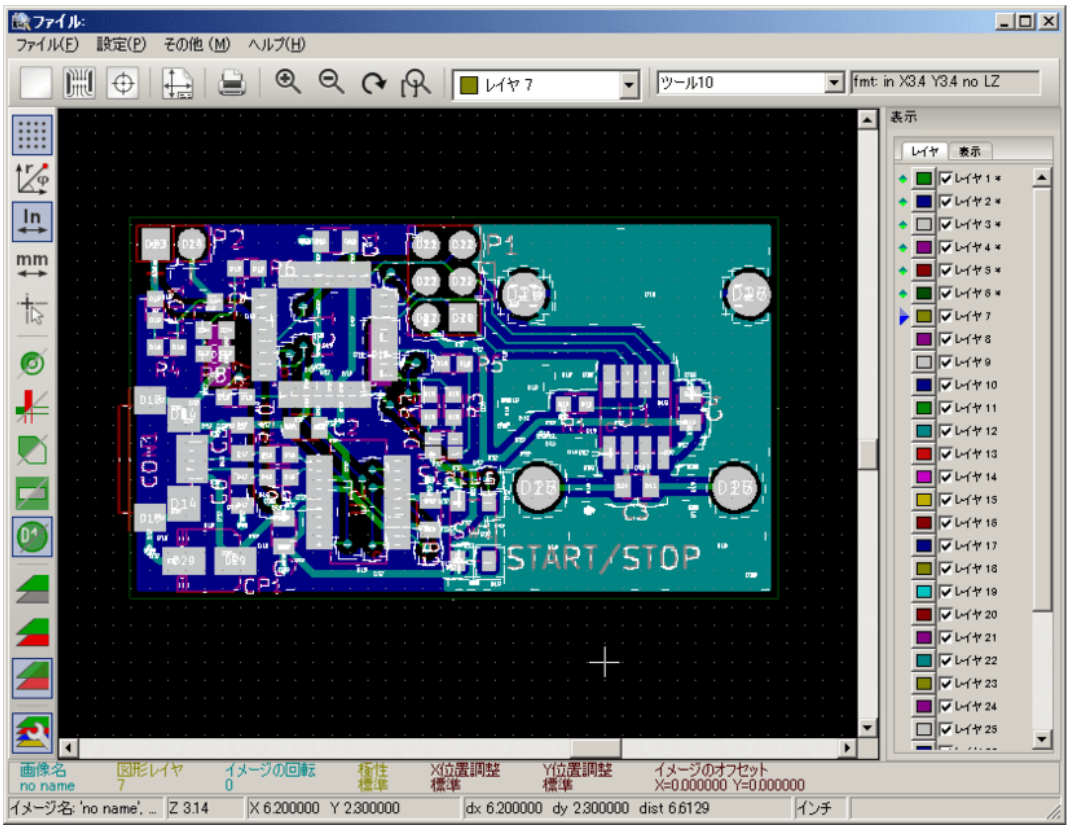
1 GerbView の紹介

GerbView は、ガーバー・ファイル (RS 274 X フォーマット) と Excellon ドリル・フ&#x

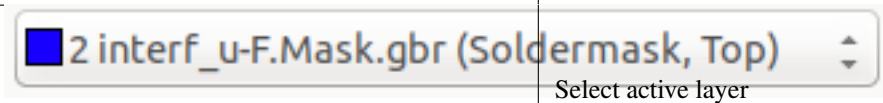
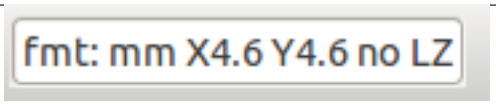
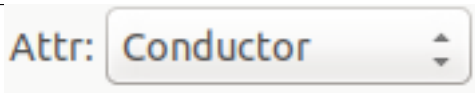
ガーバー・ファイル形式につ&#x30
the Gerber File Format Specification ドリル・ファイル&#x30fb
Excellon format description

2 インタフェース

2.1 メイン・ウィンドウ



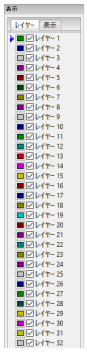
2.2 上部ツールバー

	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 左ツールバー

	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 レイヤー・マネージャー



レイヤー・マネージャーは全&#x30

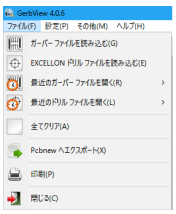
マウス・ボタンの割当て:

- 左クリック: アクティブ・レイ&#
- 右クリック: 表示/非表示/ソート&
- 中クリックまたはダブル・ク&#x3
  (色見本上): レイヤー色を選択

レイヤー・タブはレイヤーの&#x88
アイテム・タブはカラー、グ&#x3

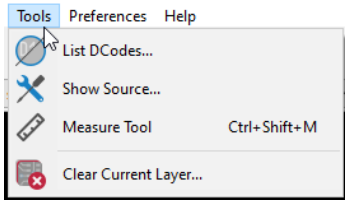
3 メニュー・バーのコ&#

3.1 ファイル・メニュー



- **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 印刷

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.
多くのフォト・プロッターは&#x