

Gerber Viewer

The KiCad Team

REVISION HISTORY			
NUMBER	DATE	DESCRIPTION	NAME

Contents

1	GerView	2
2	File	2
2.1	Open	2
2.2	Save	3
2.3	Print	4
2.4	Exit	5
3	View	5
3.1	Zoom	5
3.2	Tools menu	5
4	Help	6

リファレンス・マニュアル
著作権

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
貢献者

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.
















フィードバック

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/>

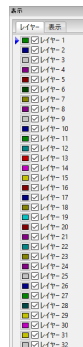
2.2 上部ツールバー

	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
 2 interf_u-F.Mask.gbr (Soldermask, Top) ▾	Select active layer
fmt: mm X4.6 Y4.6 no LZ	Display info about active layer
Cmp: C1 ▾	Highlight items belonging to selected component (Gerber X2)
Net: /DONE ▾	Highlight items belonging to selected net (Gerber X2)
Attr: Conductor ▾	Highlight items with the selected attribute (Gerber X2)
DCode: tool 10 [15.000x15.000 mils] Round ▾	Highlight items of selected D Code on the active layer

2.3

	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



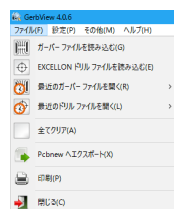
レイヤー・マネージャーは全マウス・ボタンの割当て

- 左クリック: アクティブ・レイヤージャーは全マウス・ボタンの割当て
- 右クリック: 表示/非表示/ソートャーは全マウス・ボタンの割当て
- 中クリックまたはダブル・クャはの割当て

レイヤー・タブはレイヤーの表アイテム・タブはカラー、グャはの割当て

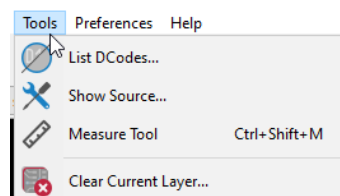
3 メニュー・バーのコャー

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



- **Dコード・リスト** は全レイヤーは全マウス・ボタンの割当て
- **Show Source** displays the Gerber file contents of the active layer in a text editor.
- **Measure Tool** allows measuring the distance between two points.
- **現在のレイヤーをクリア** はアャはの割当て

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.

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