

Gerber Viewer

The KiCad Team

REVISION HISTORY

NUMBER	DATE	DESCRIPTION	NAME

Contents

1	Gerber	2
2		2
2.1		2
2.2		3
2.3		4
2.4		5
3		6
3.1		6
3.2		6
4		7

参考手册

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.

本文档中出现的所有商标归其&#x

Contributors

KiCad 团队。

翻译人员

Jim Jiang <jim@lotlab.org>, 2018.

taotieren <admin@taotieren.com>, 2019, 2020, 2021.

Telegram 简体中文交流群; https://t.me/KiCad_zh_CN

Feedback

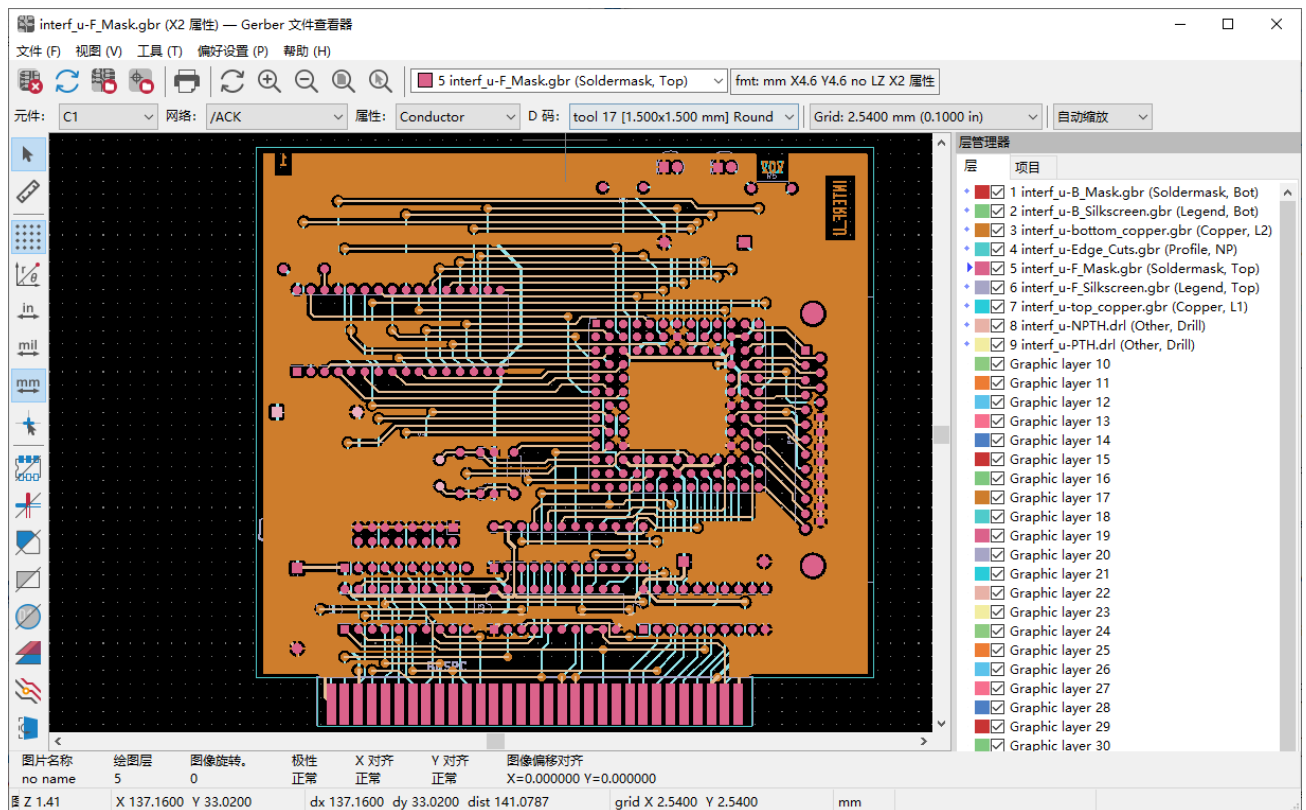
KiCad 项目欢迎与本软件或其文档相关于如何提交反馈意见或报告途
<https://www.kicad.org/help/report-an-issue/> 的说明

1 Gerber ُ览器介绍

Gerber ُ览器是一个 Gerber 文件（RS-274X 格式钻孔文件的查看器，其最多关于 Gerber 文件的更多信息，可以阅 Gerber文件格式规范 文档。关于钻孔 [http://web.archive.org/web/20071030075236/ Excellon格式描述](http://web.archive.org/web/20071030075236/Excellon格式描述) 文档。

2 界面

2.1 主窗口



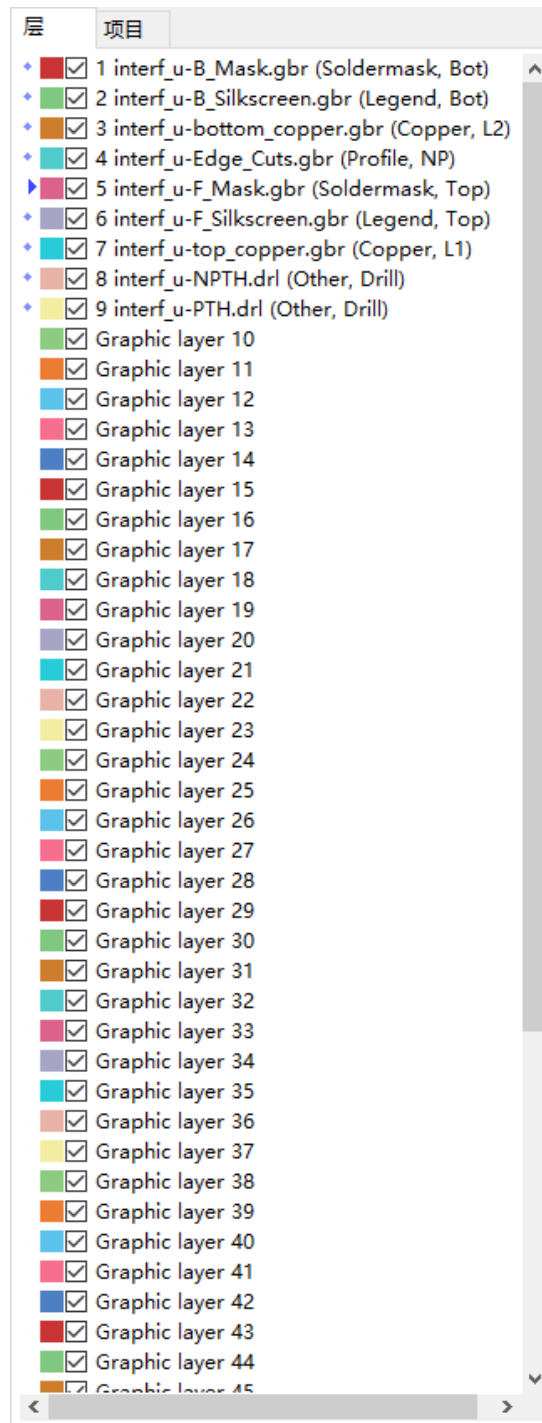
2.2 2025年5月17日 6:00;

	05;9664;6240;6709;56fe;5c42;
	52a0;8f7d; Gerber 6587;4ef6;
	52a0;8f7d; Excellon 94bb;5b54;6587;4ef6;
	8bbe;7f6e;9875;9762;5c3a;5bf8;
	6253;5370;
	91cd;7ed8;89c6;56fe;
	653e;5927;6216;7f29;5c0f;
	81ea;52a8;7f29;653e;ff08;7f29;653e;9002;&
	7f29;653e;81f3;9009;62e9;
5 interf_u-F_Mask.gbr (Soldermask, Top)	9009;62e9;6d3b;52a8;56fe;5c42;
fmt: mm X4.6 Y4.6 no LZ X2 属性	663e;793a;6709;5173;6d3b;52a8;56fe;5c42;
gerbview_x2_component_png	9ad8;4eae;663e;793a;5c5e;4e8e;6240;9009;d X2）
gerbview_x2_net_png	9ad8;4eae;663e;793a;5c5e;4e8e;6240;9009;d X2）
gerbview_x2_attributeo_png	9ad8;4eae;663e;793a;5177;6709;6240;9009;d X2）
D 码: tool 17 [1.500x1.500 mm] Round	9ad8;4eae;663e;793a;6d3b;52a8;56fe;5c42;& D码7684;9879;76ee;

2.3 左方工具栏

	选择项目
	测量两点之间
	切换网格可见性
	切换极坐标显示
	选择英寸、密尔或毫米单位
	切换全屏光标
	以草图（边框）模式显示闪灰
	以草图（边框）模式显示线极
	以草图（边框）模式显示多色
	以灰色显示负片对象
	显示/隐藏 D码
	以差异（比较）模式显示图层管理器
	在高对比度模式下显示当前将
	显示/隐藏图层管理器
	将 Gerbers 显示为镜像

2.4 2D Gerber Files



2D Gerber Files

- 2D Gerber Files
- 2D Gerber Files
- 2D Gerber Files

3 文件(F) 视图(V) 工具(T) 偏好设置(P) 帮助(H)

3.1 文件(F)



- **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 工具(T) 偏好设置(P) 帮助(H)



- **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 6253;5370;



8981;6253;5370;56fe;5c42;ff0c;8bf7;4f7f;7528; 56fe;6807;6216; 6587
→ 6253;5370; 83dc;5355;3002;

Caution



8bf7;786e;4fdd;9879;76ee;5728;53ef;6253;5370;533a;57df;5185;3002;#
6765;9009;62e9;4e00;4e2a;5408;9002;7684;9875;9762;683c;5f0f;#x3
9700;8981;6ce8;610f;7684;662f;ff0c;5927;591a;6570;7684;5149;7535;#>