

# KiCad 命令行界面

The KiCad Team

# Table of Contents

KiCad 命令行界面简介 .....	2
封装命令 .....	3
封装导出 .....	3
封装升级 .....	4
Jobset commands .....	6
PCB 命令 .....	7
PCB DRC .....	7
PCB BREP (OCCT) export .....	8
PCB 钻孔文件导出 .....	9
PCB DXF 导出 .....	10
PCB GenCAD export .....	12
PCB Gerber 导出：每个文件一层 .....	13
导出 PCB Gerber：每个文件可导出多个层 .....	15
PCB GLB 导出 .....	17
PCB IPC-2581 导出 .....	18
PCB IPC-D-356 export .....	19
PCB ODB++ export .....	20
PCB PDF 导出 .....	20
PCB PLY file export .....	22
PCB 位置文件导出 .....	24
PCB STEP 导出 .....	24
PCB STL export .....	26
PCB SVG 导出 .....	27
PCB VRML 导出 .....	29
PCB XAO export .....	30
PCB render .....	32
原理图命令 .....	34
原理图 ERC .....	34
原理图物料清单导出 .....	35
原理图 DXF 导出 .....	36
原理图 HPGL 导出 .....	37
原理图网表导出 .....	38
原理图 PDF 导出 .....	38
原理图 PostScript 导出 .....	39
原理图物料清单导出（旧版 BOM 脚本） .....	40
原理图 SVG 导出 .....	41
符号命令 .....	42
符号导出 .....	42
符号升级 .....	42
版本命令 .....	44

# *KiCad 9.0 Reference Manual*

## **Copyright**

本文档的版权归（C）2023-2024 所有，由下列贡献者提供。您可以根据 GNU 通用公共许可证（<http://www.gnu.org/licenses/gpl.html>）版本 3 或更高版本或知识共享署名许可证（<http://creativecommons.org/licenses/by/3.0/>）版本 3.0 或更高版本的条款分发和/或修改它。

本指南中的所有商标均属于其合法所有者。

## **Contributors**

Graham Keeth

## **翻译人员**

taotieren <[admin@taotieren.com](mailto:admin@taotieren.com)>, 2019-2024.

## **Feedback**

KiCad 项目欢迎与软件或其文档相关的反馈、错误报告和建议。有关如何提交反馈或报告问题的更多信息，请参阅 <https://www.kicad.org/help/report-an-issue/> 上的说明

## **Software and Documentation Version**

This user manual is based on KiCad 9.0.6. Functionality and appearance may be different in other versions of KiCad.

Documentation revision: 90da21fb.

# KiCad 命令行界面简介

KiCad 提供了一个命令行界面，可以通过运行 `kicad-cli` 二进制文件来使用。使用命令行界面，您可以以自动化方式对原理图、PCB、符号和封装执行许多操作，例如绘制 PCB 设计中的 Gerber 文件或将符号库从传统文件格式升级到现代格式。

## NOTE

在 macOS 上，`kicad-cli` 可执行文件位于 `/Applications/KiCad/KiCad.app/Contents/MacOS/kicad-cli`。

The `kicad-cli` command has 6 subcommands: `fp`, `jobset`, `pcb`, `sch`, `sym`, and `version`. Each subcommand may have its own subcommands and arguments. For example, to export Gerber files from a PCB you could run `kicad-cli pcb export gerbers example.kicad_pcb`.

您可以添加 `--help` 或 `-h` 标志来查看有关每个子命令的信息。例如，运行 `kicad-cli pcb -h` 打印有关 `pcb` 子命令的使用信息，而 `kicad-cli pcb export gerbers -h` 专门打印 `pcb export gerbers` 子命令的使用信息。

# 封装命令

fp 子命令将封装导出为另一种格式，或将封装库升级到 KiCad 封装文件格式的当前版本。

## 封装导出

fp export svg 命令将指定库中的一个或多个封装导出到 SVG 文件中。

Usage: kicad-cli fp export svg [--help] [--output OUTPUT\_DIR] [--layers LAYER\_LIST] [--define-var KEY=VALUE] [--theme VAR] [--footprint FOOTPRINT\_NAME] [--sketch-pads-on-fab-layers] [--hide-DNP-footprints-on-fab-layers] [--sketch-DNP-footprints-on-fab-layers] [--crossout-DNP-footprints-on-fab-layers] [--black-and-white] INPUT\_DIR

位置参数：

INPUT_DIR	要导出的封装库目录（.pretty）。
-----------	---------------------

可选参数：

<code>-h, --help</code>	Show help for the footprint SVG export command.
<code>-o &lt;output dir&gt;, --output &lt;output dir&gt;</code>	The output folder for the exported files. One file is output for each layer of each footprint in the library. When <code>--output</code> is not used, the files are exported to the current directory.
<code>-l &lt;layer list&gt;, --layers &lt;layer list&gt;</code>	A comma-separated list of layer names to export from the footprint, such as <code>F.Cu,B.Cu</code> . If no layers are given, all layers are exported. Layer names can be specified as canonical layer names ( <code>F.Cu</code> , <code>In.1</code> , <code>F.Fab</code> , etc.) or as user-defined (custom) layer names, but canonical layer names are matched first.
<code>-D &lt;variable name&gt;=&lt;value&gt;, --define-var &lt;variable_name&gt;=&lt;value&gt;</code>	Add or override project variable definitions. Can be used multiple times to define multiple variables.
<code>-t &lt;theme name&gt;, --theme &lt;theme name&gt;</code>	The name of the theme to use for export. If no theme is given, the footprint editor's currently selected theme is used.
<code>--fp &lt;footprint&gt;, --footprint &lt;footprint&gt;</code>	The name of the specific footprint to export from the library. When this argument is not used, all footprints in the library are exported.
<code>--sp, --sketch-pads-on-fab-layers</code>	Draw pad outlines and their numbers on front and back fab layers.
<code>--hdnp, --hide-DNP-footprints-on-fab-layers</code>	Don't plot text and graphics of DNP footprints on fab layers.
<code>--sdnp, --sketch-DNP-footprints-on-fab-layers</code>	Plot graphics of DNP footprints in sketch mode on fab layers.
<code>--cdnp, --crossout-DNP-footprints-on-fab-layers</code>	Plot an "X" over the courtyard of DNP footprints on fab layers, and strikeout their reference designators.
<code>--black-and-white</code>	Export footprints in black and white.

## 封装升级

The `fp upgrade` command converts the specified footprint library from a legacy KiCad footprint format or a non-KiCad footprint format to the native format for the current version of KiCad. If the input library is already in the current file format, no action is taken.

Supported input footprint formats are:

- KiCad footprint library ( `.pretty` folder with `.kicad_mod` files)
- KiCad (pre-5.0) footprint library ( `.mod`, `.emp` )
- Altium footprint library ( `.PcbLib` )
- Altium integrated library ( `.IntLib` )
- CADSTAR PCB archive ( `.cpa` )
- EAGLE XML library ( `.lbr` )

- EasyEDA (JLCEDA) Std file ( `.json` )
- EasyEDA (JLCEDA) Pro file ( `.elibrz` , `.epro` , `.zip` )
- GEDA/PCB library (folder with `.fp` files)

Usage: `kiCad-cli fp upgrade [--help] [--output OUTPUT_DIR] [--force] INPUT_DIR`

位置参数：

INPUT_DIR	Footprint library directory to upgrade. For KiCad format footprint libraries, this is the <code>.pretty</code> directory, not a <code>.kicad_mod</code> file.
-----------	---

可选参数：

<code>-h</code> , <code>--help</code>	Show help for the footprint upgrade command.
<code>-o &lt;output dir&gt;</code> , <code>--output &lt;output dir&gt;</code>	The output directory for the upgraded footprints. When <code>--output</code> is not used, the upgraded footprints are saved over the original footprints.
<code>--force</code>	Re-save the input library even if it is already in the current file format.

# Jobset commands

The `jobset run` command runs a predefined [jobset](#).

Usage: `kicad-cli jobset run [--help] [--stop-on-error] [--file JOB_FILE] [--output OUTPUT] INPUT_FILE`

位置参数：

INPUT_FILE	Project file to use with the jobset.
------------	--------------------------------------

可选参数：

<code>-h, --help</code>	Show help for the jobset command.
<code>--stop-on-error</code>	As jobs are executed in sequence, stop running after a job fails. If not given, jobs will continue executing after any job fails.
<code>-f &lt;jobset file&gt;, --file &lt;jobset file&gt;</code>	The jobset file ( <code>.kicad_jobset</code> ) to run.
<code>--output &lt;destination description or ID&gt;</code>	<p>The jobset destination to generate. If no destination is specified, all destinations will be generated.</p> <p>The destination is specified by its description or by its unique ID. The specified description must be unique; if the jobset contains more than one destination with the given description, none of them will be run.</p> <p>IDs are inherently unique and can be used to refer to a destination even if the destination's description is not unique. The ID for each destination is printed by the <code>jobset run</code> command when <code>--output</code> is not used. It can also be found in the <code>.kicad_jobset</code> file under the destination's <code>id</code> key.</p>



# PCB 命令

`pcb` 命令运行设计规则检查或将电路板导出为各种其他文件格式，包括制造和 3D 文件。

## PCB DRC

`pcb drc` 命令在电路板上运行设计规则检查并生成报告。

用法：`kicad-cli pcb drc [--help] [--output OUTPUT_FILE] [--define-var KEY=VALUE] [--format FORMAT] [--all-track-errors] [--schematic-parity] [--units UNITS] [--severity-all] [--severity-error] [--severity-warning] [--severity-exclusions] [--exit-code-violations] INPUT_FILE`

位置参数：

INPUT_FILE	要在其上运行 DRC 的电路板文件。
------------	--------------------

<code>-h, --help</code>	Show help for the DRC command.
<code>-o &lt;output filename&gt;, `--output &lt;output filename&gt;</code>	Output filename for the generated DRC report. When <code>--output</code> is not used, the output filename will be the same as the input file, with the <code>.rpt</code> or <code>.json</code> file extension, depending on the selected format.
<code>-D &lt;variable name&gt;=&lt;value&gt;, --define-var &lt;variable_name&gt;=&lt;value&gt;</code>	Add or override project variable definitions. Can be used multiple times to define multiple variables.
<code>--format &lt;format&gt;</code>	Report file format. Options are <code>report</code> (default) or <code>json</code> .
<code>--all-track-errors</code>	Report all errors for each track.
<code>--schematic-parity</code>	Test for parity between PCB and schematic.
<code>--units &lt;unit&gt;</code>	Units to use in the report. Options are <code>mm</code> (default), <code>in</code> , or <code>mils</code> .
<code>--severity-all</code>	Report all DRC violations. This is equivalent to using all of the other DRC severity options.
<code>--severity-error</code>	Report all error-level DRC violations. This can be combined with the other DRC severity options.
<code>--severity-warning</code>	Report all warning-level DRC violations. This can be combined with the other DRC severity options.
<code>--severity-exclusions</code>	Report all excluded DRC violations. This can be combined with the other DRC severity options.
<code>--exit-code-violations</code>	Return an exit code depending on whether or not DRC violations exist. The exit code is 0 if no violations are found, and 5 if any violations are found.

## PCB BREP (OCCT) export

The `pcb export brep` command exports a board design to a BREP (OCCT-native boundary representation) 3D model file.

```
Usage: kicad-cli pcb export brep [--help] [--output OUTPUT_FILE] [--define-var KEY=VALUE] [--force] [--no-unspecified] [--no-dnp] [--grid-origin] [--drill-origin] [--subst-models] [--board-only] [--cut-vias-in-body] [--no-board-body] [--no-components] [--component-filter VAR] [--include-tracks] [--include-pads] [--include-zones] [--include-inner-copper] [--include-silkscreen] [--include-soldermask] [--fuse-shapes] [--fill-all-vias] [--min-distance MIN_DIST] [--net-filter VAR] [--user-origin VAR] INPUT_FILE
```

位置参数：

INPUT_FILE	要导出的电路板文件。
------------	------------

可选参数：

<code>-h, --help</code>	Show help for the BREP export command.
<code>-o &lt;output filename&gt;, --output &lt;output filename&gt;</code>	The output filename. When <code>--output</code> is not used, the output filename will be the same as the input file, with the <code>.brep</code> file extension.
<code>-D &lt;variable name&gt;=&lt;value&gt;, --define-var &lt;variable_name&gt;=&lt;value&gt;</code>	Add or override project variable definitions. Can be used multiple times to define multiple variables.
<code>-f, --force</code>	Overwrite output file.
<code>--no-unspecified</code>	Exclude 3D models of components with "unspecified" footprint type.
<code>--no-dnp</code>	Exclude 3D models of components with "Do not populate" attribute.
<code>--grid-origin</code>	Use grid origin as origin of output file.
<code>--drill-origin</code>	Use drill origin as origin of output file.
<code>--subst-models</code>	Replace VRML models in footprints with STEP or IGS models of the same name, if they exist.
<code>--board-only</code>	Only include the board itself in the generated model; exclude all component models.
<code>--cut-vias-in-body</code>	Cut via holes in board body even if conductor layers are not exported.
<code>--no-board-body</code>	Exclude board body.
<code>--no-components</code>	Exclude 3D models for components.
<code>--component-filter &lt;reference designator list&gt;</code>	Only include component 3D models matching this list of reference designators (comma-separated, wildcards supported)

<code>--include-tracks</code>	Include tracks and vias on outer conductor layers in export (time consuming).
<code>--include-pads</code>	Include pads in export (time consuming).
<code>--include-zones</code>	Include zones in export (time consuming).
<code>--include-inner-copper</code>	Include elements on inner conductor layers in export.
<code>--include-silkscreen</code>	Include silkscreen graphics in export as a set of flat faces.
<code>--include-soldermask</code>	Include solder mask layers in export as a set of flat faces.
<code>--fuse-shapes</code>	Fuse overlapping geometry together in export (time consuming).
<code>--fill-all-vias</code>	Don't cut via holes in conductor layers.
<code>--min-distance &lt;min distance&gt;</code>	Tolerance for considering two points to be in the same location. Default: 0.01mm.
<code>--net-filter &lt;net filter&gt;</code>	Only include copper items belonging to nets matching this wildcard.
<code>--user-origin &lt;output origin&gt;</code>	Specify a custom origin for the output file, with X and Y coordinates. For example, 1x1in, 1x1inch, or 25.4x25.4mm. The default unit is millimeters.

## PCB 钻孔文件导出

`pcb export drill` 命令从电路板导出钻孔文件。

用法：`kicad-cli pcb export drill [--help] [--output OUTPUT_DIR] [--format FORMAT] [--drill-origin DRILL_ORIGIN] [--excellon-zeros-format ZEROS_FORMAT] [--excellon-oval-format OVAL_FORMAT] [--excellon-units UNITS] [--excellon-mirror-y] [--excellon-min-header] [--excellon-separate-th] [--generate-map] [--map-format MAP_FORMAT] [--gerber-precision VAR] INPUT_FILE`

位置参数：

INPUT_FILE	要导出的电路板文件。
------------	------------

可选参数：

<code>-h, --help</code>	Show help for the drill file export command.
<code>-o &lt;output dir&gt;, --output &lt;output dir&gt;</code>	The output directory for the drill file(s). When <code>--output</code> is not used, the drill file(s) are saved in the current directory.
<code>--format &lt;format&gt;</code>	The drill file format. Options are <code>excellon</code> (default) or <code>gerber</code> .
<code>--drill-origin &lt;origin&gt;</code>	The coordinate origin for the drill file. Options are <code>absolute</code> (default) to use the board's absolute origin or <code>plot</code> to use the board's drill/placement origin.
<code>--excellon-zeros-format &lt;format&gt;</code>	The zeros format for the drill file. Options are <code>decimal</code> (default), <code>suppressleading</code> , <code>suppresstrailing</code> , or <code>keep</code> . Only applies to Excellon format drill files.
<code>--excellon-oval-format &lt;format&gt;</code>	Control the oval holes drill mode. Options are <code>route</code> and <code>alternate</code> (default). Only applies to Excellon format drill files.
<code>-u &lt;units&gt;, --excellon-units &lt;units&gt;</code>	The units for the drill file. Options are <code>mm</code> (default) or <code>in</code> . Only applies to Excellon format drill files.
<code>--excellon-mirror-y</code>	Mirror the drill file in the Y direction. Only applies to Excellon format drill files.
<code>--excellon-min-header</code>	Use a minimal header in the drill file. Only applies to Excellon format drill files.
<code>--excellon-separate-th</code>	Generate separate drill files for plated and non-plated through holes. Only applies to Excellon format drill files.
<code>--generate-map</code>	Generate a map file in addition to the drill file.
<code>--map-format &lt;format&gt;</code>	The map file format. Options are <code>pdf</code> (default), <code>gerberx2</code> , <code>ps</code> , <code>dxf</code> , or <code>svg</code> .
<code>--gerber-precision &lt;precision&gt;</code>	The precision (number of digits) for the drill file. Valid options are <code>5</code> or <code>6</code> (default). Only applies to Gerber format drill files.

## PCB DXF 导出

`pcb export dxf` 命令将电路板设计导出为 DXF 文件。

Usage: `kicad-cli pcb export dxf [--help] [--output OUTPUT_FILE] [--layers LAYER_LIST] [--drawing-sheet SHEET_PATH] [--define-var KEY=VALUE] [--exclude-refdes] [--exclude-value] [--sketch-pads-on-fab-layers] [--hide-DNP-footprints-on-fab-layers] [--sketch-DNP-footprints-on-fab-layers] [--crossout-DNP-footprints-on-fab-layers] [--subtract-soldermask] [--use-contours] [--use-drill-origin] [--include-border-title] [--output-units UNITS] [--drill-shape-opt VAR] [--common-layers COMMON_LAYER_LIST] [--mode-single] [--mode-multi] [--plot-invisible-text] INPUT_FILE`

位置参数：

INPUT_FILE	要导出的电路板文件。
------------	------------

可选参数：

<code>-h, --help</code>	Show help for the DXF export command.
<code>-o &lt;output dir&gt;, --output &lt;output dir&gt;</code>	The output folder or filename for the exported files. When <code>--mode-single</code> is used, this is the output filename. If <code>--output</code> is not used, the output filename will be the same as the input file, with the <code>.pdf</code> file extension. When <code>--mode-multi</code> is used, this is the output directory. If <code>--output</code> is not used, the files are exported to the current directory.
<code>-l &lt;layer list&gt;, --layers &lt;layer list&gt;</code>	A comma-separated list of layer names to export from the footprint, such as <code>F.Cu,B.Cu</code> . At least one layer must be given. Layer names can be specified as canonical layer names ( <code>F.Cu</code> , <code>In.1</code> , <code>F.Fab</code> , etc.) or as user-defined (custom) layer names, but canonical layer names are matched first.
<code>--drawing-sheet &lt;sheet path&gt;</code>	Path to drawing sheet to use in plot, overriding the drawing sheet specified in the board file.
<code>-D &lt;variable name&gt;=&lt;value&gt;, --define-var &lt;variable_name&gt;=&lt;value&gt;</code>	Add or override project variable definitions. Can be used multiple times to define multiple variables.
<code>--erd, --exclude-refdes</code>	Exclude footprint reference designators from plot.
<code>--ev, --exclude-value</code>	Exclude footprint values from plot.
<code>--sp, --sketch-pads-on-fab-layers</code>	Draw pad outlines and their numbers on front and back fab layers.
<code>--hdnp, --hide-DNP-footprints-on-fab-layers</code>	Don't plot text and graphics of DNP footprints on fab layers.
<code>--sdnp, --sketch-DNP-footprints-on-fab-layers</code>	Plot graphics of DNP footprints in sketch mode on fab layers.
<code>--cdnp, --crossout-DNP-footprints-on-fab-layers</code>	Plot an "X" over the courtyard of DNP footprints on fab layers, and strikeout their reference designators.
<code>--subtract-soldermask</code>	Remove silkscreen from areas without soldermask.
<code>--uc, --use-contours</code>	Plot graphic items using their contours.
<code>--udo, --use-drill-origin</code>	Plot using the drill/place file origin.
<code>-ibt, --include-border-title</code>	Include sheet border and title block in plot.
<code>--ou &lt;unit&gt;, --output-units &lt;unit&gt;</code>	Output units. Options are <code>mm</code> or <code>in</code> (default).
<code>--drill-shape-opt &lt;shape&gt;</code>	The shape of drill marks in the plot. Options are <code>0</code> for no drill marks, <code>1</code> for small marks, or <code>2</code> for actual size marks (default).

<code>--cl &lt;layer list&gt;, --common-layers &lt;layer list&gt;</code>	A comma-separated list of layer names to plot on all layers, such as F.Cu,B.Cu. Layer names can be specified as canonical layer names ( F.Cu , In.1 , F.Fab , etc.) or as user-defined (custom) layer names, but canonical layer names are matched first.
<code>--mode-single</code>	Generates a single file with the output arg path acting as the complete directory and filename path. <code>COMMON_LAYER_LIST</code> does not function in this mode. Instead <code>LAYER_LIST</code> controls all layers plotted.
<code>--mode-multi</code>	Plot the layers to one or more DXF files, with each file representing a single layer from <code>LAYER_LIST</code> . The output path specifies the directory in which the files will be written.
<code>--plot-invisible-text</code>	Force plotting of values and references, even if they are invisible. This argument is deprecated as of KiCad 9.0.1 and has no effect. It will be removed in a future version of KiCad. To plot invisible text, edit the board so that the text is no longer invisible.

## PCB GenCAD export

The `pcb export gencad` command exports a board design to a GenCAD file.

Usage: `kicad-cli pcb export gencad [--help] [--output OUTPUT_FILE] [--define-var KEY=VALUE] [--flip-bottom-pads] [--unique-pins] [--unique-footprints] [--use-drill-origin] [--store-origin-coord] INPUT_FILE`

位置参数：

INPUT_FILE	要导出的电路板文件。
------------	------------

可选参数：

<code>-h, --help</code>	Show help for the DXF export command.
<code>-o &lt;output filename&gt;, --output &lt;output filename&gt;</code>	The output filename. When <code>--output</code> is not used, the output filename will be the same as the input file, with the <code>.cad</code> file extension.
<code>-D &lt;variable name&gt;=&lt;value&gt;, --define-var &lt;variable_name&gt;=&lt;value&gt;</code>	Add or override project variable definitions. Can be used multiple times to define multiple variables.
<code>-f, --flip-bottom-pads</code>	Flip bottom footprint padstacks.
<code>--unique-pins</code>	Generate unique pin names.
<code>--unique-footprints</code>	Generate a new shape for each footprint instance (do not reuse shapes).
<code>--use-drill-origin</code>	Use drill/place file origin as origin.
<code>--store-origin-coord</code>	Save the origin coordinates in the file.

# PCB Gerber 导出：每个文件一层

pcb export gerbers 命令将电路板设计导出到 Gerber 文件，每个文件一层。

NOTE

请注意有两种不同的 Gerber 导出命令，即 gerber 和 gerbers。gerber 命令将多个 PCB 层绘制到单个 Gerber 文件中，而 gerbers 命令将多个 Gerber 文件绘制到单个 Gerber 文件中。在制作 PCB 时，通常应使用 gerbers 命令。

```
Usage: kicad-cli pcb export gerbers [--help] [--output OUTPUT_DIR] [--layers LAYER_LIST] [--drawing-sheet SHEET_PATH] [--define-var KEY=VALUE] [--exclude-refdes] [--exclude-value] [--include-border-title] [--sketch-pads-on-fab-layers] [--hide-DNP-footprints-on-fab-layers] [--sketch-DNP-footprints-on-fab-layers] [--crossout-DNP-footprints-on-fab-layers] [--no-x2] [--no-netlist] [--subtract-soldermask] [--disable-aperture-macros] [--use-drill-file-origin] [--precision PRECISION] [--no-protel-ext] [--plot-invisible-text] [--common-layers COMMON_LAYER_LIST] [--board-plot-params] INPUT_FILE
```

位置参数：

INPUT_FILE	要导出的电路板文件。
------------	------------

可选参数：

-h, --help	Show help for the Gerber export command.
-o <output dir>, --output <output dir>	The output folder for the exported files. One file is output for each layer. When --output is not used, the files are exported to the current directory.
-l <layer list>, --layers <layer list>	A comma-separated list of layer names to plot from the board, such as F.Cu,B.Cu. If this argument is not used, all layers will be plotted. A seperate output file is plotted for each layer. Layer names can be specified as canonical layer names ( F.Cu , In.1 , F.Fab , etc.) or as user-defined (custom) layer names, but canonical layer names are matched first.
--drawing-sheet <sheet path>	Path to drawing sheet to use in plot, overriding the drawing sheet specified in the board file.
-D <variable name>=<value>, --define-var <variable_name>=<value>	Add or override project variable definitions. Can be used multiple times to define multiple variables.
--erd, --exclude-refdes	Exclude footprint reference designators from plot.
--ev, --exclude-value	Exclude footprint values from plot.
--ibt, --include-border-title	Include the sheet border and title block.

<code>--sp, --sketch-pads-on-fab-layers</code>	Draw pad outlines and their numbers on front and back fab layers.
<code>--hdnp, --hide-DNP-footprints-on-fab-layers</code>	Don't plot text and graphics of DNP footprints on fab layers.
<code>--sdnp, --sketch-DNP-footprints-on-fab-layers</code>	Plot graphics of DNP footprints in sketch mode on fab layers.
<code>--cdnp, --crossout-DNP-footprints-on-fab-layers</code>	Plot an "X" over the courtyard of DNP footprints on fab layers, and strikeout their reference designators.
<code>--no-x2</code>	Do not use the extended X2 format.
<code>--no-netlist</code>	Do not include netlist attributes.
<code>--subtract-soldermask</code>	Remove silkscreen from areas without soldermask.
<code>--disable-aperture-macros</code>	Disable aperture macros.
<code>--use-drill-file-origin</code>	Use drill/place file origin instead of absolute origin.
<code>--precision &lt;precision&gt;</code>	The precision (number of digits) for the Gerber files. Valid options are 5 or 6 (default).
<code>--no-protel-ext</code>	Use <code>.gbr</code> file extension instead of Protel file extensions ( <code>.gbl</code> , <code>.gtl</code> , etc.).
<code>--plot-invisible-text</code>	Force plotting of values and references, even if they are invisible. This argument is deprecated as of KiCad 9.0.1 and has no effect. It will be removed in a future version of KiCad. To plot invisible text, edit the board so that the text is no longer invisible.



<code>--cl &lt;layer list&gt;, --common-layers &lt;layer list&gt;</code>	A comma-separated list of layer names to plot on all layers, such as <code>F.Cu,B.Cu</code> . Each layer specified is included in every output file. Layer names can be specified as canonical layer names ( <code>F.Cu</code> , <code>In.1</code> , <code>F.Fab</code> , etc.) or as user-defined (custom) layer names, but canonical layer names are matched first.
<code>--board-plot-params</code>	Use the Gerber plot settings already configured in the board file.

## 导出 PCB Gerber：每个文件可导出多个层

`pcb export gerber` 命令将一个或多个板层导出到单个 Gerber 文件。

### NOTE

请注意有两种不同的 Gerber 导出命令，即 `gerber` 和 `gerbers`。`gerber` 命令将多个 PCB 层绘制到单个 Gerber 文件中，而 `gerbers` 命令将多个 Gerber 文件绘制到单个 Gerber 文件中。在制作 PCB 时，通常应使用 `gerbers` 命令。

### WARNING

The `pcb export gerber` command is deprecated in KiCad 9.0 and will be removed in KiCad 10.0. Please use the `pcb export gerbers` command instead.

Usage: `kicad-cli pcb export gerber [--help] [--output OUTPUT_FILE] [--layers LAYER_LIST] [--drawing-sheet SHEET_PATH] [--define-var KEY=VALUE] [--exclude-refdes] [--exclude-value] [--include-border-title] [--sketch-pads-on-fab-layers] [--hide-DNP-footprints-on-fab-layers] [--sketch-DNP-footprints-on-fab-layers] [--crossout-DNP-footprints-on-fab-layers] [--no-x2] [--no-netlist] [--subtract-soldermask] [--disable-aperture-macros] [--use-drill-file-origin] [--precision PRECISION] [--no-protel-ext] [--plot-invisible-text] INPUT_FILE`

位置参数：

<code>INPUT_FILE</code>	要导出的电路板文件。
-------------------------	------------

可选参数：

<code>-h, --help</code>	Show help for the Gerber export command.
<code>-o &lt;output filename&gt;, --output &lt;output filename&gt;</code>	The output filename. When <code>--output</code> is not used, the output filename will be the same as the input file, with the <code>.gbr</code> file extension.
<code>-l &lt;layer list&gt;, --layers &lt;layer list&gt;</code>	A comma-separated list of layer names to plot from the board, such as <code>F.Cu,B.Cu</code> . All layers will be plotted in the output file. At least one layer must be given. Layer names can be specified as canonical layer names ( <code>F.Cu</code> , <code>In.1</code> , <code>F.Fab</code> , etc.) or as user-defined (custom) layer names, but canonical layer names are matched first.
<code>--drawing-sheet &lt;sheet path&gt;</code>	Path to drawing sheet to use in plot, overriding the drawing sheet specified in the board file.

<code>-D &lt;variable name&gt;=&lt;value&gt;, --define-var &lt;variable_name&gt;=&lt;value&gt;</code>	Add or override project variable definitions. Can be used multiple times to define multiple variables.
<code>--erd, --exclude-refdes</code>	Exclude footprint reference designators from plot.
<code>--ev, --exclude-value</code>	Exclude footprint values from plot.
<code>--ibt, --include-border-title</code>	Include the sheet border and title block.
<code>--sp, --sketch-pads-on-fab-layers</code>	Draw pad outlines and their numbers on front and back fab layers.
<code>--hdnp, --hide-DNP-footprints-on-fab-layers</code>	Don't plot text and graphics of DNP footprints on fab layers.
<code>--sdnp, --sketch-DNP-footprints-on-fab-layers</code>	Plot graphics of DNP footprints in sketch mode on fab layers.
<code>--cdnp, --crossout-DNP-footprints-on-fab-layers</code>	Plot an "X" over the courtyard of DNP footprints on fab layers, and strikeout their reference designators.
<code>--no-x2</code>	Do not use the extended X2 format.
<code>--no-netlist</code>	Do not include netlist attributes.
<code>--subtract-soldermask</code>	Remove silkscreen from areas without soldermask.
<code>--disable-aperture-macros</code>	Disable aperture macros.
<code>--use-drill-file-origin</code>	Use drill/place file origin instead of absolute origin.
<code>--precision &lt;precision&gt;</code>	The precision (number of digits) for the Gerber files. Valid options are 5 or 6 (default).
<code>--no-protel-ext</code>	Use <code>.gbr</code> file extension instead of Protel file extensions ( <code>.gbl</code> , <code>.gtl</code> , etc.).
<code>--plot-invisible-text</code>	Force plotting of values and references, even if they are invisible. This argument is deprecated as of KiCad 9.0.1 and has no effect. It will be removed in a future version of KiCad. To plot invisible text, edit the board so that the text is no longer invisible.

## PCB GLB 导出

The `pcb export glb` command exports a board design to a GLB (binary glTF) 3D model file.

```
Usage: kicad-cli pcb export glb [--help] [--output OUTPUT_FILE] [--define-var KEY=VALUE] [--force] [--no-unspecified] [--no-dnp] [--grid-origin] [--drill-origin] [--subst-models] [--board-only] [--cut-vias-in-body] [--no-board-body] [--no-components] [--component-filter VAR] [--include-tracks] [--include-pads] [--include-zones] [--include-inner-copper] [--include-silkscreen] [--include-soldermask] [--fuse-shapes] [--fill-all-vias] [--min-distance MIN_DIST] [--net-filter VAR] [--user-origin VAR] INPUT_FILE
```

位置参数：

INPUT_FILE	要导出的电路板文件。
------------	------------

可选参数：

<code>-h, --help</code>	Show help for the GLB export command.
<code>-o &lt;output filename&gt;, --output &lt;output filename&gt;</code>	The output filename. When <code>--output</code> is not used, the output filename will be the same as the input file, with the <code>.glb</code> file extension.
<code>-D &lt;variable name&gt;=&lt;value&gt;, --define-var &lt;variable_name&gt;=&lt;value&gt;</code>	Add or override project variable definitions. Can be used multiple times to define multiple variables.
<code>-f, --force</code>	Overwrite output file.
<code>--no-unspecified</code>	Exclude 3D models of components with "unspecified" footprint type.
<code>--no-dnp</code>	Exclude 3D models of components with "Do not populate" attribute.
<code>--grid-origin</code>	Use grid origin as origin of output file.
<code>--drill-origin</code>	Use drill origin as origin of output file.
<code>--subst-models</code>	Replace VRML models in footprints with STEP or IGS models of the same name, if they exist.
<code>--board-only</code>	Only include the board itself in the generated model; exclude all component models.
<code>--cut-vias-in-body</code>	Cut via holes in board body even if conductor layers are not exported.
<code>--no-board-body</code>	Exclude board body.
<code>--no-components</code>	Exclude 3D models for components.
<code>--component-filter &lt;reference designator list&gt;</code>	Only include component 3D models matching this list of reference designators (comma-separated, wildcards supported)

<code>--include-tracks</code>	Include tracks and vias on outer conductor layers in export (time consuming).
<code>--include-pads</code>	Include pads in export (time consuming).
<code>--include-zones</code>	Include zones in export (time consuming).
<code>--include-inner-copper</code>	Include elements on inner conductor layers in export.
<code>--include-silkscreen</code>	Include silkscreen graphics in export as a set of flat faces.
<code>--include-soldermask</code>	Include solder mask layers in export as a set of flat faces.
<code>--fuse-shapes</code>	Fuse overlapping geometry together in export (time consuming).
<code>--fill-all-vias</code>	Don't cut via holes in conductor layers.
<code>--min-distance &lt;min distance&gt;</code>	Tolerance for considering two points to be in the same location. Default: 0.01mm.
<code>--net-filter &lt;net filter&gt;</code>	Only include copper items belonging to nets matching this wildcard.
<code>--user-origin &lt;output origin&gt;</code>	Specify a custom origin for the output file, with X and Y coordinates. For example, 1x1in, 1x1inch, or 25.4x25.4mm. The default unit is millimeters.

## PCB IPC-2581 导出

`pcb export ipc2581` 命令以 IPC-2581 格式导出电路板设计。

用法： `kicad-cli pcb export ipc2581 [--help] [--output OUTPUT_FILE] [--drawing-sheet SHEET_PATH] [--define-var KEY=VALUE] [--precision PRECISION] [--compress] [--version VAR] [--units VAR] [--bom-col-int-id FIELD_NAME] [--bom-col-mfg-pn FIELD_NAME] [--bom-col-mfg FIELD_NAME] [--bom-col-dist-pn FIELD_NAME] [--bom-col-dist FIELD_NAME] INPUT_FILE`

INPUT_FILE	要导出的电路板文件。
------------	------------

可选参数：

<code>-h, --help</code>	Show help for the IPC-2581 export command.
<code>-o &lt;output filename&gt;, --output &lt;output filename&gt;</code>	The output filename. When <code>--output</code> is not used, the output filename will be the same as the input file, with the <code>.xml</code> file extension.
<code>--drawing-sheet &lt;sheet path&gt;</code>	Path to drawing sheet to use in plot, overriding the drawing sheet specified in the board file.
<code>-D &lt;variable name&gt;=&lt;value&gt;, --define-var &lt;variable_name&gt;=&lt;value&gt;</code>	Add or override project variable definitions. Can be used multiple times to define multiple variables.
<code>--precision &lt;precision&gt;</code>	The precision (number of digits after the decimal separator) for the exported file. The default is 6.
<code>--compress</code>	Compress output file as a ZIP file.
<code>--version &lt;IPC-2581 standard version&gt;</code>	IPC-2581 standard version to use. Options are <code>B</code> or <code>C</code> (default).
<code>--units</code>	Units to use in export. Options are <code>mm</code> (default) or <code>in</code> .
<code>--bom-col-int-id</code>	Name of the part field to use for the Bill of Materials Internal ID column. This can be any footprint field, or blank to omit this column.
<code>--bom-col-mfg-pn</code>	Name of the part field to use for the Bill of Materials Manufacturer Part Number column. This can be any footprint field, or blank to omit this column.
<code>--bom-col-mfg</code>	Name of the part field to use for the Bill of Materials Manufacturer column. This can be any footprint field, or blank to omit this column.
<code>--bom-col-dist-pn</code>	Name of the part field to use for the Bill of Materials Distributor Part Number column. This can be any footprint field, or blank to omit this column.
<code>--bom-col-dist</code>	Name of the part field to use for the Bill of Materials Distributor column. This can be any footprint field, or blank to omit this column.

## PCB IPC-D-356 export

The `pcb export ipcd356` command generates an IPC-D-356 netlist from the board design.

Usage: `kiCad-cli pcb export ipcd356 [--help] [--output OUTPUT_FILE] INPUT_FILE`

位置参数：

<code>INPUT_FILE</code>	要导出的电路板文件。
-------------------------	------------

可选参数：

<code>-h, --help</code>	Show help for the IPC-D-356 export command.
<code>-o &lt;output filename&gt;, --output &lt;output filename&gt;</code>	The output filename. When <code>--output</code> is not used, the output filename will be the same as the input file, with the <code>.d356</code> file extension.

## PCB ODB++ export

The `pcb export odb` command exports a board design in ODB++ format.

Usage: `kicad-cli pcb export odb [--help] [--output OUTPUT_FILE] [--drawing-sheet SHEET_PATH] [--define-var KEY=VALUE] [--precision PRECISION] [--compression VAR] [--units VAR] INPUT_FILE`

位置参数：

INPUT_FILE	要导出的电路板文件。
------------	------------

可选参数：

<code>-h, --help</code>	Show help for the ODB++ export command.
<code>-o &lt;output filename&gt;, --output &lt;output filename&gt;</code>	The output filename, or folder name if no compression is used.
<code>--drawing-sheet &lt;sheet path&gt;</code>	Path to drawing sheet to use in plot, overriding the drawing sheet specified in the board file.
<code>-D &lt;variable name&gt;=&lt;value&gt;, --define-var &lt;variable_name&gt;=&lt;value&gt;</code>	Add or override project variable definitions. Can be used multiple times to define multiple variables.
<code>--precision &lt;precision&gt;</code>	The precision (number of digits after the decimal separator) for the exported file. The default is 2.
<code>--compression &lt;mode&gt;</code>	Compression mode. Options are <code>none</code> , <code>zip</code> (default), or <code>tgz</code> .
<code>--units &lt;unit&gt;</code>	Units to use in the output file. Options are <code>mm</code> (default) or <code>in</code> .

## PCB PDF 导出

The `pcb export pdf` command exports a board design to a PDF file. Each layer can be plotted as its own file or as a sheet within a single file.

Usage: `kicad-cli pcb export pdf [--help] [--output OUTPUT_FILE] [--layers LAYER_LIST] [--drawing-sheet SHEET_PATH] [--define-var KEY=VALUE] [--mirror] [--exclude-refdes] [--exclude-value] [--include-border-title] [--subtract-soldermask] [--sketch-pads-on-fab-layers] [--hide-DNP-footprints-on-fab-layers] [--sketch-DNP-footprints-on-fab-layers] [--crossout-DNP-footprints-on-fab-layers] [--negative] [--black-and-white] [--theme THEME_NAME] [--drill-shape-opt VAR] [--common-layers COMMON_LAYER_LIST] [--plot-invisible-text] [--mode-single] [--mode-separate] [--mode-multipage] INPUT_FILE`

位置参数：

INPUT_FILE	要导出的电路板文件。
------------	------------

可选参数：

<code>-h, --help</code>	Show help for the PDF export command.
<code>-o &lt;output dir&gt;, --output &lt;output dir&gt;</code>	The output folder or filename for the exported files. When <code>--mode-single</code> or <code>--mode-multipage</code> is used, this is the output filename. If this argument is not used, the output filename will be the same as the input file, with the <code>.pdf</code> file extension. When <code>--mode-separate</code> is used, this is the output directory. If <code>--output</code> is not used, the files are exported to the current directory.
<code>-l &lt;layer list&gt;, --layers &lt;layer list&gt;</code>	A comma-separated list of layer names to export from the board, such as <code>F.Cu,B.Cu</code> . At least one layer must be given. Layer names can be specified as canonical layer names ( <code>F.Cu</code> , <code>In.1</code> , <code>F.Fab</code> , etc.) or as user-defined (custom) layer names, but canonical layer names are matched first.
<code>--drawing-sheet &lt;sheet path&gt;</code>	Path to drawing sheet to use in plot, overriding the drawing sheet specified in the board file.
<code>-D &lt;variable name&gt;=&lt;value&gt;, --define-var &lt;variable_name&gt;=&lt;value&gt;</code>	Add or override project variable definitions. Can be used multiple times to define multiple variables.
<code>-m, --mirror</code>	Mirror the board. This can be useful for showing bottom layers.
<code>--erd, --exclude-refdes</code>	Exclude footprint reference designators from plot.
<code>--ev, --exclude-value</code>	Exclude footprint values from plot.
<code>--ibt, --include-border-title</code>	Include the sheet border and title block.
<code>--subtract-soldermask</code>	Remove silkscreen from areas without soldermask.
<code>--sp, --sketch-pads-on-fab-layers</code>	Draw pad outlines and their numbers on front and back fab layers.
<code>--hdnp, --hide-DNP-footprints-on-fab-layers</code>	Don't plot text and graphics of DNP footprints on fab layers.
<code>--sdnp, --sketch-DNP-footprints-on-fab-layers</code>	Plot graphics of DNP footprints in sketch mode on fab layers.
<code>--cdnp, --crossout-DNP-footprints-on-fab-layers</code>	Plot an "X" over the courtyard of DNP footprints on fab layers, and strikeouts their reference designators.
<code>-n, --negative</code>	Plot in negative.
<code>--black-and-white</code>	Plot in black and white.

<code>-t &lt;theme name&gt;, --theme &lt;theme name&gt;</code>	The name of the theme to use for export. If no theme is given, the board editor's currently selected theme is used.
<code>--drill-shape-opt</code>	The shape of drill marks in the plot. Options are 0 for no drill marks, 1 for small marks, or 2 for actual size marks (default).
<code>--cl &lt;layer list&gt;, --common-layers &lt;layer list&gt;</code>	A comma-separated list of layer names to plot on all layers, such as <code>F.Cu,B.Cu</code> . Layer names can be specified as canonical layer names ( <code>F.Cu</code> , <code>In.1</code> , <code>F.Fab</code> , etc.) or as user-defined (custom) layer names, but canonical layer names are matched first.
<code>--plot-invisible-text</code>	Force plotting of values and references, even if they are invisible. This argument is deprecated as of KiCad 9.0.1 and has no effect. It will be removed in a future version of KiCad. To plot invisible text, edit the board so that the text is no longer invisible.
<code>--mode-single</code>	Generates a single file with the output arg path acting as the complete directory and filename path. <code>COMMON_LAYER_LIST</code> does not function in this mode. Instead <code>LAYER_LIST</code> controls all layers plotted. All specified layers are plotted on a single page.
<code>--mode-separate</code>	Plot the layers to one or more PDF files, with each file representing a single layer from <code>LAYER_LIST</code> . The output path specifies the directory in which the files will be written.
<code>--mode-multipage</code>	Plot the layers to a single PDF file with multiple pages, with each page representing a single layer from <code>LAYER_LIST</code> . The output path specifies the complete directory and filename path.

## PCB PLY file export

The `pcb export ply` command exports a board design to a PLY 3D model file.

Usage: `kicad-cli pcb export ply [--help] [--output OUTPUT_FILE] [--define-var KEY=VALUE] [--force] [--no-unspecified] [--no-dnp] [--grid-origin] [--drill-origin] [--subst-models] [--board-only] [--cut-vias-in-body] [--no-board-body] [--no-components] [--component-filter VAR] [--include-tracks] [--include-pads] [--include-zones] [--include-inner-copper] [--include-silkscreen] [--include-soldermask] [--fuse-shapes] [--fill-all-vias] [--min-distance MIN_DIST] [--net-filter VAR] [--user-origin VAR] INPUT_FILE`

位置参数：

<code>INPUT_FILE</code>	要导出的电路板文件。
-------------------------	------------

可选参数：

<code>-h, --help</code>	Show help for the PLY export command.
<code>-o &lt;output filename&gt;, --output &lt;output filename&gt;</code>	The output filename. When <code>--output</code> is not used, the output filename will be the same as the input file, with the <code>.ply</code> file extension.



<code>-D &lt;variable name&gt;= &lt;value&gt;, --define-var &lt;variable_name&gt;=&lt;value&gt;</code>	Add or override project variable definitions. Can be used multiple times to define multiple variables.
<code>-f, --force</code>	Overwrite output file.
<code>--no-unspecified</code>	Exclude 3D models of components with "unspecified" footprint type.
<code>--no-dnp</code>	Exclude 3D models of components with "Do not populate" attribute.
<code>--grid-origin</code>	Use grid origin as origin of output file.
<code>--drill-origin</code>	Use drill origin as origin of output file.
<code>--subst-models</code>	Replace VRML models in footprints with STEP or IGS models of the same name, if they exist.
<code>--board-only</code>	Only include the board itself in the generated model; exclude all component models.
<code>--cut-vias-in-body</code>	Cut via holes in board body even if conductor layers are not exported.
<code>--no-board-body</code>	Exclude board body.
<code>--no-components</code>	Exclude 3D models for components.
<code>--component-filter &lt;reference designator list&gt;</code>	Only include component 3D models matching this list of reference designators (comma-separated, wildcards supported)
<code>--include-tracks</code>	Include tracks and vias on outer conductor layers in export (time consuming).
<code>--include-pads</code>	Include pads in export (time consuming).
<code>--include-zones</code>	Include zones in export (time consuming).
<code>--include-inner-copper</code>	Include elements on inner conductor layers in export.
<code>--include-silkscreen</code>	Include silkscreen graphics in export as a set of flat faces.
<code>--include-soldermask</code>	Include solder mask layers in export as a set of flat faces.
<code>--fuse-shapes</code>	Fuse overlapping geometry together in export (time consuming).
<code>--fill-all-vias</code>	Don't cut via holes in conductor layers.
<code>--min-distance &lt;min distance&gt;</code>	Tolerance for considering two points to be in the same location. Default: 0.01mm.
<code>--net-filter &lt;net filter&gt;</code>	Only include copper items belonging to nets matching this wildcard.
<code>--user-origin &lt;output origin&gt;</code>	Specify a custom origin for the output file, with X and Y coordinates. For example, 1x1in, 1x1inch, or 25.4x25.4mm. The default unit is millimeters.

# PCB 位置文件导出

pcb export pos 命令从电路板设计中导出位置文件。

用法： kicad-cli pcb export pos [--help] [--output OUTPUT\_FILE] [--side VAR] [--format FORMAT] [--units UNITS] [--bottom-negate-x] [--use-drill-file-origin] [--smd-only] [--exclude-fp-th] [--exclude-dnp] [--gerber-board-edge] INPUT\_FILE

位置参数：

INPUT_FILE	要导出的电路板文件。
------------	------------

可选参数：

-h, --help	Show help for the position file export command.
-o <output filename>, --output <output filename>	The output filename. When --output is not used, the output filename will be the same as the input file, with the .pos file extension.
--side <side>	The side of the board to export. Options are front, back, or both (default). Gerber format does not support both.
--format <format>	The position file format. Options are ascii (default), csv, or gerber.
--units <unit>	Units for position file. Options are in (default) or mm. This option has no effect for Gerber format.
--bottom-negate-x	Use negative X coordinates for footprints on the bottom layer. This option has no effect for Gerber format.
--use-drill-file-origin	Use drill/place file origin instead of absolute origin. This option has no effect for Gerber format.
--smd-only	Include only surface-mount components. This option has no effect for Gerber format.
--exclude-fp-th	Exclude all footprints with through-hole pads. This option has no effect for Gerber format.
--exclude-dnp	Exclude all footprints with "Do not populate" attribute.
--gerber-board-edge	Include board edge layer in export (Gerber format only).

# PCB STEP 导出

pcb export step 命令将电路板设计导出为 STEP 文件。

Usage: kicad-cli pcb export step [--help] [--output OUTPUT\_FILE] [--define-var KEY=VALUE] [--force] [--no-unspecified] [--no-dnp] [--grid-origin] [--drill-origin] [--subst-models] [--board-only] [--cut-vias-in-body] [--no-board-body] [--no-components] [--component-filter VAR] [--include-tracks] [--include-pads] [--include-zones] [--include-inner-copper] [--include-

```
silkscreen] [--include-soldermask] [--fuse-shapes] [--fill-all-vias] [--min-distance MIN_DIST]
[--net-filter VAR] [--no-optimize-step] [--user-origin VAR] INPUT_FILE
```

位置参数：

INPUT_FILE	要导出的电路板文件。
------------	------------

可选参数：

-h, --help	Show help for the STEP file export command.
-o <output filename>, --output <output filename>	The output filename. When --output is not used, the output filename will be the same as the input file, with the .step file extension.
-D <variable name>=<value>, --define-var <variable_name>=<value>	Add or override project variable definitions. Can be used multiple times to define multiple variables.
-f, --force	Overwrite output file.
--no-unspecified	Exclude 3D models of components with "unspecified" footprint type.
--no-dnp	Exclude 3D models of components with "Do not populate" attribute.
--grid-origin	Use grid origin as origin of output file.
--drill-origin	Use drill origin as origin of output file.
--subst-models	Replace VRML models in footprints with STEP or IGS models of the same name, if they exist.
--board-only	Only include the board itself in the generated model; exclude all component models.
--cut-vias-in-body	Cut via holes in board body even if conductor layers are not exported.
--no-board-body	Exclude board body.
--no-components	Exclude 3D models for components.
--component-filter <reference designator list>	Only include component 3D models matching this list of reference designators (comma-separated, wildcards supported)
--include-tracks	Include tracks and vias on outer conductor layers in export (time consuming).
--include-pads	Include pads in export (time consuming).
--include-zones	Include zones in export (time consuming).
--include-inner-copper	Include elements on inner conductor layers in export.
--include-silkscreen	Include silkscreen graphics in export as a set of flat faces.

<code>--include-soldermask</code>	Include solder mask layers in export as a set of flat faces.
<code>--fuse-shapes</code>	Fuse overlapping geometry together in export (time consuming).
<code>--fill-all-vias</code>	Don't cut via holes in conductor layers.
<code>--min-distance &lt;min distance&gt;</code>	Tolerance for considering two points to be in the same location. Default: 0.01mm.
<code>--net-filter &lt;net filter&gt;</code>	Only include copper items belonging to nets matching this wildcard.
<code>--no-optimize-step</code>	Do not optimize STEP file. This enables writing parametric curves, which reduces file sizes and write/read times, but may reduce compatibility with other software.
<code>--user-origin &lt;output origin&gt;</code>	Specify a custom origin for the output file, with X and Y coordinates. For example, 1x1in, 1x1inch, or 25.4x25.4mm. The default unit is millimeters.

## PCB STL export

The `pcb export stl` command exports a board design to an STL 3D model file.

Usage: `kicad-cli pcb export stl [--help] [--output OUTPUT_FILE] [--define-var KEY=VALUE] [--force] [--no-unspecified] [--no-dnp] [--grid-origin] [--drill-origin] [--subst-models] [--board-only] [--cut-vias-in-body] [--no-board-body] [--no-components] [--component-filter VAR] [--include-tracks] [--include-pads] [--include-zones] [--include-inner-copper] [--include-silkscreen] [--include-soldermask] [--fuse-shapes] [--fill-all-vias] [--min-distance MIN_DIST] [--net-filter VAR] [--user-origin VAR] INPUT_FILE`

位置参数：

INPUT_FILE	要导出的电路板文件。
------------	------------

可选参数：

<code>-h, --help</code>	Show help for the STL export command.
<code>-o &lt;output filename&gt;, --output &lt;output filename&gt;</code>	The output filename. When <code>--output</code> is not used, the output filename will be the same as the input file, with the <code>.stl</code> file extension.
<code>-D &lt;variable name&gt;=&lt;value&gt;, --define-var &lt;variable_name&gt;=&lt;value&gt;</code>	Add or override project variable definitions. Can be used multiple times to define multiple variables.
<code>-f, --force</code>	Overwrite output file.
<code>--no-unspecified</code>	Exclude 3D models of components with "unspecified" footprint type.
<code>--no-dnp</code>	Exclude 3D models of components with "Do not populate" attribute.

<code>--grid-origin</code>	Use grid origin as origin of output file.
<code>--drill-origin</code>	Use drill origin as origin of output file.
<code>--subst-models</code>	Replace VRML models in footprints with STEP or IGS models of the same name, if they exist.
<code>--board-only</code>	Only include the board itself in the generated model; exclude all component models.
<code>--cut-vias-in-body</code>	Cut via holes in board body even if conductor layers are not exported.
<code>--no-board-body</code>	Exclude board body.
<code>--no-components</code>	Exclude 3D models for components.
<code>--component-filter</code> <reference designator list>	Only include component 3D models matching this list of reference designators (comma-separated, wildcards supported)
<code>--include-tracks</code>	Include tracks and vias on outer conductor layers in export (time consuming).
<code>--include-pads</code>	Include pads in export (time consuming).
<code>--include-zones</code>	Include zones in export (time consuming).
<code>--include-inner-copper</code>	Include elements on inner conductor layers in export.
<code>--include-silkscreen</code>	Include silkscreen graphics in export as a set of flat faces.
<code>--include-soldermask</code>	Include solder mask layers in export as a set of flat faces.
<code>--fuse-shapes</code>	Fuse overlapping geometry together in export (time consuming).
<code>--fill-all-vias</code>	Don't cut via holes in conductor layers.
<code>--min-distance</code> <min distance>	Tolerance for considering two points to be in the same location. Default: 0.01mm.
<code>--net-filter</code> <net filter>	Only include copper items belonging to nets matching this wildcard.
<code>--user-origin</code> <output origin>	Specify a custom origin for the output file, with X and Y coordinates. For example, 1x1in, 1x1inch, or 25.4x25.4mm. The default unit is millimeters.

## PCB SVG 导出

`pcb export svg` 命令将电路板设计导出为 SVG 文件。

Usage: `kicad-cli pcb export svg` [`--help`] [`--output` OUTPUT\_FILE] [`--layers` LAYER\_LIST] [`--drawing-sheet` SHEET\_PATH] [`--define-var` KEY=VALUE] [`--subtract-soldermask`] [`--mirror`] [`--theme` THEME\_NAME] [`--negative`] [`--black-and-white`] [`--sketch-pads-on-fab-layers`] [`--hide-DNP-footprints-on-fab-layers`] [`--sketch-DNP-footprints-on-fab-layers`] [`--crossout-DNP-footprints-`

```
on-fab-layers] [--page-size-mode MODE] [--fit-page-to-board] [--exclude-drawing-sheet] [--drill-shape-opt SHAPE_OPTION] [--common-layers COMMON_LAYER_LIST] [--mode-single] [--mode-multi] [--plot-invisible-text] INPUT_FILE
```

位置参数：

INPUT_FILE	要导出的电路板文件。
------------	------------

可选参数：

-h, --help	Show help for the SVG file export command.
-o <output dir>, --output <output dir>	The output folder or filename for the exported files. When --mode-single is used, this is the output filename. If --output is not used, the output filename will be the same as the input file, with the .pdf file extension. When --mode-multi is used, this is the output directory. If --output is not used, the files are exported to the current directory.
-l <layer list>, --layers <layer list>	A comma-separated list of layer names to export from the board, such as F.Cu,B.Cu. At least one layer must be given. Layer names can be specified as canonical layer names ( F.Cu, In.1, F.Fab, etc.) or as user-defined (custom) layer names, but canonical layer names are matched first.
--drawing-sheet <sheet path>	Path to drawing sheet to use in plot, overriding the drawing sheet specified in the board file.
-D <variable name>=<value>, --define-var <variable_name>=<value>	Add or override project variable definitions. Can be used multiple times to define multiple variables.
--subtract-soldermask	Remove silkscreen from areas without soldermask.
-m, --mirror	Mirror the board. This can be useful for showing bottom layers.
-t <theme name>, --theme <theme name>	The name of the theme to use for export. If no theme is given, the board editor's currently selected theme is used.
-n, --negative	Plot in negative.
--black-and-white	Plot in black and white.
--sp, --sketch-pads-on-fab-layers	Draw pad outlines and their numbers on front and back fab layers.
--hdnp, --hide-DNP-footprints-on-fab-layers	Don't plot text and graphics of DNP footprints on fab layers.
--sdnp, --sketch-DNP-footprints-on-fab-layers	Plot graphics of DNP footprints in sketch mode on fab layers.

<code>--cdnp, --crossout-DNP-footprints-on-fab-layers</code>	Plot an "X" over the courtyard of DNP footprints on fab layers, and strikeouts their reference designators.
<code>--page-size-mode &lt;mode&gt;</code>	Set page sizing mode. Options are 0 (default), 1, or 2. 0 sets the output page size to fit the entire sheet, including drawing sheet frame and title block. 1 sets the output page size to match the current page size. 2 sets the output page size to the size of the board itself.
<code>--fit-page-to-board</code>	Set the SVG size to match the board outline. This is equivalent to <code>--page-size-mode 2</code> .
<code>--exclude-drawing-sheet</code>	Plot SVG without a drawing sheet.
<code>--drill-shape-opt</code>	The shape of drill marks in the plot. Options are 0 for no drill marks, 1 for small marks, or 2 for actual size marks (default).
<code>--cl &lt;layer list&gt;, --common-layers &lt;layer list&gt;</code>	A comma-separated list of layer names to plot on all layers, such as F.Cu,B.Cu. Layer names can be specified as canonical layer names (F.Cu, In.1, F.Fab, etc.) or as user-defined (custom) layer names, but canonical layer names are matched first.
<code>--mode-single</code>	Generates a single file with the output arg path acting as the complete directory and filename path. COMMON_LAYER_LIST does not function in this mode. Instead LAYER_LIST controls all layers plotted.
<code>--mode-multi</code>	Plot the layers to one or more SVG files, with each file representing a single layer from LAYER_LIST. The output path specifies the directory in which the files will be written.
<code>--plot-invisible-text</code>	Force plotting of values and references, even if they are invisible. This argument is deprecated as of KiCad 9.0.1 and has no effect. It will be removed in a future version of KiCad. To plot invisible text, edit the board so that the text is no longer invisible.

## PCB VRML 导出

The `pcb export vrml` command exports a board design to a VRML 3D model file.

Usage: `kicad-cli pcb export vrml [--help] [--output OUTPUT_FILE] [--define-var KEY=VALUE] [--force] [--no-unspecified] [--no-dnp] [--user-origin VAR] [--units VAR] [--models-dir VAR] [--models-relative] INPUT_FILE`

位置参数：

INPUT_FILE	要导出的电路板文件。
------------	------------

可选参数：

<code>-h, --help</code>	Show help for the VRML export command.
<code>-o &lt;output filename&gt;, --output &lt;output filename&gt;</code>	The output filename. When <code>--output</code> is not used, the output filename will be the same as the input file, with the <code>.wr1</code> file extension.
<code>-D &lt;variable name&gt;=&lt;value&gt;, --define-var &lt;variable_name&gt;=&lt;value&gt;</code>	Add or override project variable definitions. Can be used multiple times to define multiple variables.
<code>-f, --force</code>	Overwrite output file.
<code>--no-unspecified</code>	Exclude 3D models of components with "unspecified" footprint type.
<code>--no-dnp</code>	Exclude 3D models of components with "Do not populate" attribute.
<code>--user-origin &lt;output origin&gt;</code>	Specify a custom origin for the output file, with X and Y coordinates. For example, <code>1x1in</code> , <code>1x1inch</code> , or <code>25.4x25.4mm</code> . The default unit is millimeters. If this option is not given, the board center is used.
<code>--units &lt;units&gt;</code>	Units to use in the output file. Options are <code>mm</code> , <code>m</code> , <code>in</code> (default), or <code>tenths</code> (tenths of an inch).
<code>--models-dir &lt;output model directory&gt;</code>	Name of output directory to copy component models into. If not used, component models are embedded into the output file.
<code>--models-relative</code>	With <code>--models-dir</code> , use relative paths in the output file.

## PCB XAO export

The `pcb export xao` command exports a board design to an XAO (SALOME/Gmsh) 3D model file.

Usage: `kicad-cli pcb export xao [--help] [--output OUTPUT_FILE] [--define-var KEY=VALUE] [--force] [--no-unspecified] [--no-dnp] [--grid-origin] [--drill-origin] [--subst-models] [--board-only] [--cut-vias-in-body] [--no-board-body] [--no-components] [--component-filter VAR] [--include-tracks] [--include-pads] [--include-zones] [--include-inner-copper] [--include-silkscreen] [--include-soldermask] [--fuse-shapes] [--fill-all-vias] [--min-distance MIN_DIST] [--net-filter VAR] [--user-origin VAR] INPUT_FILE`

位置参数：

INPUT_FILE	要导出的电路板文件。
------------	------------

可选参数：

<code>-h, --help</code>	Show help for the XAO export command.
<code>-o &lt;output filename&gt;, --output &lt;output filename&gt;</code>	The output filename. When <code>--output</code> is not used, the output filename will be the same as the input file, with the <code>.xao</code> file extension.
<code>-D &lt;variable name&gt;=&lt;value&gt;, --define-var &lt;variable_name&gt;=&lt;value&gt;</code>	Add or override project variable definitions. Can be used multiple times to define multiple variables.



<code>-f, --force</code>	Overwrite output file.
<code>--no-unspecified</code>	Exclude 3D models of components with "unspecified" footprint type.
<code>--no-dnp</code>	Exclude 3D models of components with "Do not populate" attribute.
<code>--grid-origin</code>	Use grid origin as origin of output file.
<code>--drill-origin</code>	Use drill origin as origin of output file.
<code>--subst-models</code>	Replace VRML models in footprints with STEP or IGS models of the same name, if they exist.
<code>--board-only</code>	Only include the board itself in the generated model; exclude all component models.
<code>--cut-vias-in-body</code>	Cut via holes in board body even if conductor layers are not exported.
<code>--no-board-body</code>	Exclude board body.
<code>--no-components</code>	Exclude 3D models for components.
<code>--component-filter</code> <code>&lt;reference designator</code> <code>list&gt;</code>	Only include component 3D models matching this list of reference designators (comma-separated, wildcards supported)
<code>--include-tracks</code>	Include tracks and vias on outer conductor layers in export (time consuming).
<code>--include-pads</code>	Include pads in export (time consuming).
<code>--include-zones</code>	Include zones in export (time consuming).
<code>--include-inner-copper</code>	Include elements on inner conductor layers in export.
<code>--include-silkscreen</code>	Include silkscreen graphics in export as a set of flat faces.
<code>--include-soldermask</code>	Include solder mask layers in export as a set of flat faces.
<code>--fuse-shapes</code>	Fuse overlapping geometry together in export (time consuming).
<code>--fill-all-vias</code>	Don't cut via holes in conductor layers.
<code>--min-distance</code> <code>&lt;min</code> <code>distance&gt;</code>	Tolerance for considering two points to be in the same location. Default: 0.01mm.
<code>--net-filter</code> <code>&lt;net filter&gt;</code>	Only include copper items belonging to nets matching this wildcard.
<code>--user-origin</code> <code>&lt;output</code> <code>origin&gt;</code>	Specify a custom origin for the output file, with X and Y coordinates. For example, 1x1in, 1x1inch, or 25.4x25.4mm. The default unit is millimeters.

## PCB render

The `pcb render` command generates a raytraced rendering of the 3D model of the board and saves it to a PNG or JPEG file.

Usage: `kicad-cli pcb render [--help] [--output OUTPUT_FILE] [--define-var KEY=VALUE] [--width WIDTH] [--height HEIGHT] [--side SIDE] [--background BG] [--quality QUALITY] [--preset PRESET] [--floor] [--perspective] [--zoom ZOOM] [--pan VECTOR] [--pivot PIVOT] [--rotate ANGLES] [--light-top COLOR] [--light-bottom COLOR] [--light-side COLOR] [--light-camera COLOR] [--light-side-elevation ANGLE] INPUT_FILE`

位置参数：

INPUT_FILE	Board file to render.
------------	-----------------------

可选参数：

<code>-h, --help</code>	Show help for the render command.
<code>-o &lt;output filename&gt;, --output &lt;output filename&gt;</code>	The output filename. This argument must be given. The file extension given in this argument determines the output image file format. The filename must end with either <code>.png</code> (for PNG files) or <code>.jpg</code> / <code>.jpeg</code> (for JPG files).
<code>-D &lt;variable name&gt;=&lt;value&gt;, --define-var &lt;variable_name&gt;=&lt;value&gt;</code>	Add or override project variable definitions. Can be used multiple times to define multiple variables.
<code>-w &lt;width&gt;, --width &lt;width&gt;</code>	Image width in pixels. Default: 1600.
<code>-h &lt;height&gt;, --height &lt;height&gt;</code>	Image height in pixels. Default: 900.
<code>--side &lt;side&gt;</code>	The side of the board to render. Options are <code>top</code> (default), <code>bottom</code> , <code>left</code> , <code>right</code> , <code>front</code> , or <code>back</code> .
<code>--background &lt;background&gt;</code>	Image background. Options are <code>default</code> (default), <code>transparent</code> , or <code>opaque</code> . For PNG files, <code>default</code> is <code>transparent</code> . For JPG files, <code>default</code> is <code>opaque</code> .
<code>--quality &lt;quality&gt;</code>	Render quality. Options are <code>basic</code> (default), <code>high</code> , <code>user</code> . When <code>user</code> is specified, the render settings stored in the project are used.
<code>--preset &lt;color preset&gt;</code>	Color preset. Options are <code>follow_pcb_editor</code> , <code>follow_plot_settings</code> (default), or <code>legacy_preset_flag</code> .
<code>--floor</code>	Enables floor, shadows and post-processing, even if disabled in quality preset.
<code>--perspective</code>	Use perspective projection instead of orthogonal.

<code>--zoom &lt;zoom level&gt;</code>	Camera zoom factor as an integer. Default: 1.
<code>--pan &lt;camera pan&gt;</code>	Set camera pan location, in millimeters, with the format 'X,Y,Z' , e.g. '3,0,0' .
<code>--pivot &lt;pivot&gt;</code>	Set pivot point relative to the board center in centimeters, with the format 'X,Y,Z' e.g. '-10,2,0' .
<code>--rotate &lt;rotation&gt;</code>	Set board rotation around pivot point, in degrees, with the format 'X,Y,Z' , e.g. '-45,0,45' for isometric view.
<code>--light-top &lt;intensity&gt;</code>	Top light intensity, format 'R,G,B' or a single number, range: 0-1.
<code>--light-bottom &lt;intensity&gt;</code>	Bottom light intensity, format 'R,G,B' or a single number, range: 0-1.
<code>--light-side &lt;intensity&gt;</code>	Side lights intensity, format 'R,G,B' or a single number, range: 0-1.
<code>--light-camera &lt;intensity&gt;</code>	Camera light intensity, format 'R,G,B' or a single number, range: 0-1.
<code>--light-side-elevation &lt;elevation&gt;</code>	Side lights elevation angle in degrees, range: 0-90.

# 原理图命令

`sch` 命令用于运行电气规则检查，将原理图导出为其他各种文件格式，或导出物料清单或网表。每个子命令都有自己的选项。

## 原理图 ERC

`sch erc` 命令对原理图运行电气规则检查并生成报告。

用法：`kicad-cli sch erc [--help] [--output OUTPUT_FILE] [--define-var KEY=VALUE] [--format VAR] [--units VAR] [--severity-all] [--severity-error] [--severity-warning] [--severity-exclusions] [--exit-code-violations] INPUT_FILE`

位置参数：

INPUT_FILE	运行 ERC 的原理图文件。
------------	----------------

可选参数：

<code>-h, --help</code>	Show help for the ERC command.
<code>-o &lt;output filename&gt;, --output &lt;output filename&gt;</code>	Output filename for the generated ERC report. When <code>--output</code> is not used, the output filename will be the same as the input file, with the <code>.rpt</code> or <code>.json</code> file extension, depending on the selected format.
<code>-D &lt;variable name&gt;=&lt;value&gt;, --define-var &lt;variable_name&gt;=&lt;value&gt;</code>	Add or override project variable definitions. Can be used multiple times to define multiple variables.
<code>--format &lt;format&gt;</code>	Report file format. Options are <code>report</code> (default) or <code>json</code> .
<code>--units &lt;unit&gt;</code>	Units to use in the report. Options are <code>mm</code> (default), <code>in</code> , or <code>mils</code> .
<code>--severity-all</code>	Report all ERC violations. This is equivalent to using all of the other ERC severity options.
<code>--severity-error</code>	Report all error-level ERC violations. This can be combined with the other ERC severity options.
<code>--severity-warning</code>	Report all warning-level ERC violations. This can be combined with the other ERC severity options.
<code>--severity-exclusions</code>	Report all excluded ERC violations. This can be combined with the other ERC severity options.
<code>--exit-code-violations</code>	Return an exit code depending on whether or not ERC violations exist. The exit code is 0 if no violations are found, and 5 if any violations are found.

# 原理图物料清单导出

`sch export bom` 命令从原理图中导出 BOM。BOM 导出具有许多用于控制格式和包含字段的选项。此导出方法等同于 [导出 BOM](#) 从符号字段表。

NOTE

要使用旧版 XML 和 Python BOM 脚本工作流导出 BOM，请使用 `sch export python-bom` 命令。

```
Usage: kicad-cli sch export bom [--help] [--output OUTPUT_FILE] [--preset PRESET] [--format-
preset FMT_PRESET] [--fields FIELDS] [--labels LABELS] [--group-by GROUP_BY] [--sort-field
SORT_BY] [--sort-asc] [--filter FILTER] [--exclude-dnp] [--include-excluded-from-bom] [--field-
delimiter FIELD_DELIM] [--string-delimiter STR_DELIM] [--ref-delimiter REF_DELIM] [--ref-range-
delimiter REF_RANGE_DELIM] [--keep-tabs] [--keep-line-breaks] INPUT_FILE
```

位置参数：

INPUT_FILE	要导出的原理图文件。
------------	------------

可选参数：

<code>-h, --help</code>	Shows help message and exits
<code>-o &lt;output filename&gt;, --output &lt;output filename&gt;</code>	The output filename. When <code>--output</code> is not used, the output filename will be the same as the input file, with a <code>.csv</code> file extension.
<code>--preset &lt;preset&gt;</code>	Use a named BOM preset setting from the schematic, e.g. "Grouped By Value".
<code>--format-preset &lt;format preset&gt;</code>	Use a named BOM format preset setting from the schematic, e.g. CSV.
<code>--fields &lt;fields&gt;</code>	An ordered list of fields to export. <code>*</code> includes all fields. Special symbol fields such as DNP or Exclude from board can be accessed with <code>\${DNP}</code> or <code>\${EXCLUDE_FROM_BOARD}</code> , respectively (see the <a href="#">text variable documentation</a> for a list of fields). Default: "Reference,Value,Footprint,\${QUANTITY},\${DNP}".
<code>--labels &lt;labels&gt;</code>	An ordered list of labels to apply the exported fields (default: "Refs,Value,Footprint,Qty,DNP").
<code>--group-by &lt;fields&gt;</code>	Fields to group references by when field values match.
<code>--sort-field &lt;fields&gt;</code>	Field name to sort by (default: "Reference").
<code>--sort-asc</code>	If given, sort in ascending order. If not given, sort in descending order.
<code>--filter &lt;filter&gt;</code>	Filter string to remove output lines.
<code>--exclude-dnp</code>	Exclude symbols with the "Do not populate" attribute.
<code>--include-excluded-from-bom</code>	Include symbols marked "Exclude from BOM".
<code>--field-delimiter &lt;delimiter&gt;</code>	Separator between output fields/columns (default: ", ").
<code>--string-delimiter &lt;delimiter&gt;</code>	Character to surround fields with (none by default).
<code>--ref-delimiter &lt;delimiter&gt;</code>	Character to place between individual references (default: ", ").
<code>--ref-range-delimiter &lt;delimiter&gt;</code>	Character to place in ranges of references (default: "-"). Leave blank for no ranges.
<code>--keep-tabs</code>	Keep tab characters from input fields. Stripped by default.
<code>--keep-line-breaks</code>	Keep line break characters from input fields. Stripped by default.

## 原理图 DXF 导出

`sch export dxf` 命令将原理图导出到 DXF 文件。设计中的每个图纸都导出到其自己的文件中。

Usage: `kicad-cli sch export dxf [--help] [--output OUTPUT_DIR] [--drawing-sheet SHEET_PATH] [--define-var KEY=VALUE] [--theme THEME_NAME] [--black-and-white] [--exclude-drawing-sheet] [--default-font VAR] [--pages PAGE_LIST] INPUT_FILE`

位置参数：

INPUT_FILE	要导出的原理图文件。
------------	------------

可选参数：

<code>-h, --help</code>	Show help for the DXF file export command.
<code>-o &lt;output dir&gt;, --output &lt;output dir&gt;</code>	The output folder for the exported files. One file is output for each sheet. When <code>--output</code> is not used, the files are exported to the current directory.
<code>--drawing-sheet &lt;sheet path&gt;</code>	Path to drawing sheet to use in plot, overriding the drawing sheet specified in the schematic file.
<code>-D &lt;variable name&gt;=&lt;value&gt;, --define-var &lt;variable_name&gt;=&lt;value&gt;</code>	Add or override project variable definitions. Can be used multiple times to define multiple variables.
<code>-t &lt;theme name&gt;, --theme &lt;theme name&gt;</code>	The name of the theme to use for export. If no theme is given, the schematic editor's currently selected theme is used.
<code>-b, --black-and-white</code>	Export schematic in black and white.
<code>-e, --exclude-drawing-sheet</code>	Plot DXF without a drawing sheet.
<code>--default-font &lt;font name&gt;</code>	Default font name. Default: "KiCad Font".
<code>-p &lt;page list&gt;, --pages &lt;page list&gt;</code>	Comma-separated list of pages to export. Blank or unspecified means all pages. To plot specific pages, give the root sheet as <code>INPUT_FILE</code> and specify the desired output pages with the <code>--pages</code> argument.

## 原理图 HPGL 导出

`sch export hpgl` 命令将原理图导出到笔式绘图仪的 HPGL 文件中。设计中的每个图纸都导出到其自己的文件中。

Usage: `kicad-cli sch export hpgl [--help] [--output OUTPUT_DIR] [--drawing-sheet SHEET_PATH] [--define-var KEY=VALUE] [--exclude-drawing-sheet] [--default-font VAR] [--pages PAGE_LIST] [--pen-size PEN_SIZE] [--origin ORIGIN] INPUT_FILE`

位置参数：

INPUT_FILE	要导出的原理图文件。
------------	------------

可选参数：

<code>-h, --help</code>	Show help for the HPGL file export command.
<code>-o &lt;output dir&gt;, --output &lt;output dir&gt;</code>	The output folder for the exported files. When this argument is not used, the files are exported to the current directory.
<code>--drawing-sheet &lt;sheet path&gt;</code>	Path to drawing sheet to use in plot, overriding the drawing sheet specified in the schematic file.
<code>-D &lt;variable name&gt;=&lt;value&gt;, --define-var &lt;variable_name&gt;=&lt;value&gt;</code>	Add or override project variable definitions. Can be used multiple times to define multiple variables.
<code>-e, --exclude-drawing-sheet</code>	Plot HPGL without a drawing sheet.
<code>--default-font &lt;font name&gt;</code>	Default font name. Default: "KiCad Font".
<code>--pages &lt;page list&gt;</code>	Comma-separated list of pages to export. Blank or unspecified means all pages. To plot specific pages, give the root sheet as <code>INPUT_FILE</code> and specify the desired output pages with the <code>--pages</code> argument.
<code>-p &lt;pen size&gt;, --pen-size &lt;pen size&gt;</code>	Set the pen width. The default pen size is 0.5 mm.
<code>-r &lt;origin&gt;, --origin &lt;origin&gt;</code>	Set plotter origin and scale. Options are 0, 1 (default), 2, or 3. 0 sets the origin to the bottom left and uses plotter units. 1 sets the origin to the center and uses plotter units. 2 scales to the page, and 3 scales to the content within the page.

## 原理图网表导出

`sch export netlist` 命令以 [各种格式](#) 格式导出网表。

用法： `kicad-cli sch export netlist [--help] [--output OUTPUT_FILE] [--format FORMAT] INPUT_FILE`

位置参数：

<code>INPUT_FILE</code>	要导出的原理图文件。
-------------------------	------------

可选参数：

<code>-h, --help</code>	Show help for the netlist export command.
<code>-o &lt;output filename&gt;, --output &lt;output filename&gt;</code>	The output filename. When <code>--output</code> is not used, the output filename will be the same as the input file, with a <code>.net</code> file extension.
<code>-f &lt;format&gt;, --format &lt;format&gt;</code>	The netlist output format. Options are <code>kicadsexpr</code> (default), <code>kicadxml</code> , <code>cadstar</code> , <code>orcadpcb2</code> , <code>spice</code> , <code>spicemodel</code> , <code>pads</code> , or <code>allegro</code> .

## 原理图 PDF 导出

`sch export pdf` 命令将原理图导出为 PDF 文件。设计中的每张图纸都会导出到 PDF 文件中的单独页面。



Usage: `kicad-cli sch export pdf [--help] [--output OUTPUT_FILE] [--drawing-sheet SHEET_PATH] [--define-var KEY=VALUE] [--theme THEME_NAME] [--black-and-white] [--exclude-drawing-sheet] [--default-font VAR] [--exclude-pdf-property-popups] [--exclude-pdf-hierarchical-links] [--exclude-pdf-metadata] [--no-background-color] [--pages PAGE_LIST] INPUT_FILE`

位置参数：

INPUT_FILE	要导出的原理图文件。
------------	------------

可选参数：

<code>-h, --help</code>	Show help for the PDF file export command.
<code>-o &lt;output filename&gt;, --output &lt;output filename&gt;</code>	The output filename. When <code>--output</code> is not used, the output filename will be the same as the input file, with a <code>.pdf</code> file extension.
<code>--drawing-sheet &lt;sheet path&gt;</code>	Path to drawing sheet to use in plot, overriding the drawing sheet specified in the schematic file.
<code>-D &lt;variable name&gt;=&lt;value&gt;, --define-var &lt;variable_name&gt;=&lt;value&gt;</code>	Add or override project variable definitions. Can be used multiple times to define multiple variables.
<code>-t &lt;theme name&gt;, --theme &lt;theme name&gt;</code>	The name of the theme to use for export. If no theme is given, the schematic editor's currently selected theme is used.
<code>-b, --black-and-white</code>	Export schematic in black and white.
<code>-e, --exclude-drawing-sheet</code>	Plot PDF without a drawing sheet.
<code>--default-font &lt;font name&gt;</code>	Default font name. Default: "KiCad Font".
<code>--exclude-pdf-property-popups</code>	Do not generate property popups in PDF.
<code>--exclude-pdf-hierarchical-links</code>	Do not generate clickable links for hierarchical elements in PDF.
<code>--exclude-pdf-metadata</code>	Do not generate PDF metadata from AUTHOR and SUBJECT variables.
<code>-n, --no-background-color</code>	Export schematic without a background color, regardless of theme.
<code>-p &lt;page list&gt;, --pages &lt;page list&gt;</code>	Comma-separated list of pages to export. Blank or unspecified means all pages. To plot specific pages, give the root sheet as <code>INPUT_FILE</code> and specify the desired output pages with the <code>--pages</code> argument.

## 原理图 PostScript 导出

`sch export ps` 命令将原理图导出到 PostScript 文件。设计中的每个图纸都导出到其自己的文件中。

Usage: `kicad-cli sch export ps [--help] [--output OUTPUT_DIR] [--drawing-sheet SHEET_PATH] [--define-var KEY=VALUE] [--theme THEME_NAME] [--black-and-white] [--exclude-drawing-sheet] [--`

default-font VAR] [--no-background-color] [--pages PAGE\_LIST] INPUT\_FILE

位置参数：

INPUT_DIR	要导出的原理图文件。
-----------	------------

可选参数：

-h, --help	Show help for the PS file export command.
-o <output dir>, --output <output dir>	The output folder for the exported files. One file is output for each sheet. When --output is not used, the files are exported to the current directory.
--drawing-sheet <sheet path>	Path to drawing sheet to use in plot, overriding the drawing sheet specified in the schematic file.
-D <variable name>=<value>, --define-var <variable_name>=<value>	Add or override project variable definitions. Can be used multiple times to define multiple variables.
-t <theme name>, --theme <theme name>	The name of the theme to use for export. If no theme is given, the schematic editor's currently selected theme is used.
-b, --black-and-white	Export schematic in black and white.
-e, --exclude-drawing-sheet	Plot PS without a drawing sheet.
--default-font <font name>	Default font name. Default: "KiCad Font".
-n, --no-background-color	Export schematic without a background color, regardless of theme.
-p <page list>, --pages <page list>	Comma-separated list of pages to export. Blank or unspecified means all pages. To plot specific pages, give the root sheet as INPUT_FILE and specify the desired output pages with the --pages argument.

## 原理图物料清单导出（旧版 BOM 脚本）

sch export python-bom 命令从原理图中导出XML BOM文件。然后，可以使用自定义脚本或外部参照中描述的脚本之一将 XML BOM 文件处理为所需的 BOM 格式：[原理图 BOM 导出文档](#)。

用法：kicad-cli sch export python-bom [--help] [--output OUTPUT\_FILE] INPUT\_FILE

位置参数：

INPUT_FILE	要导出的原理图文件。
------------	------------

可选参数：

<code>-h, --help</code>	Show help for the BOM export command.
<code>-o &lt;output filename&gt;, --output &lt;output filename&gt;</code>	The output filename. When <code>--output</code> is not used, the output filename will be the same as the input file, with a <code>-bom.xml</code> suffix and file extension.

## 原理图 SVG 导出

`sch export svg` 命令将原理图导出为 SVG 文件。设计中的每个图纸都导出到其自己的文件中。

Usage: `kicad-cli sch export svg [--help] [--output OUTPUT_DIR] [--drawing-sheet SHEET_PATH] [--define-var KEY=VALUE] [--theme THEME_NAME] [--black-and-white] [--exclude-drawing-sheet] [--default-font VAR] [--no-background-color] [--pages PAGE_LIST] INPUT_FILE`

位置参数：

INPUT_FILE	要导出的原理图文件。
------------	------------

可选参数：

<code>-h, --help</code>	Show help for the SVG file export command.
<code>-o &lt;output dir&gt;, --output &lt;output dir&gt;</code>	The output folder for the exported files. When <code>--output</code> is not used, the files are exported to the current directory.
<code>--drawing-sheet &lt;sheet path&gt;</code>	Path to drawing sheet to use in plot, overriding the drawing sheet specified in the schematic file.
<code>-D &lt;variable name&gt;=&lt;value&gt;, --define-var &lt;variable_name&gt;=&lt;value&gt;</code>	Add or override project variable definitions. Can be used multiple times to define multiple variables.
<code>-t &lt;theme name&gt;, --theme &lt;theme name&gt;</code>	The name of the theme to use for export. If no theme is given, the schematic editor's currently selected theme is used.
<code>-b, --black-and-white</code>	Export schematic in black and white.
<code>-e, --exclude-drawing-sheet</code>	Plot SVG without a drawing sheet.
<code>--default-font &lt;font name&gt;</code>	Default font name. Default: "KiCad Font".
<code>-n, --no-background-color</code>	Export schematic without a background color, regardless of theme.
<code>-p &lt;page list&gt;, --pages &lt;page list&gt;</code>	Comma-separated list of pages to export. Blank or unspecified means all pages. To plot specific pages, give the root sheet as <code>INPUT_FILE</code> and specify the desired output pages with the <code>--pages</code> argument.

# 符号命令

`sym` 子命令将符号导出为另一种格式，或将符号库升级到 KiCad 符号文件格式的当前版本。

## 符号导出

`sym export svg` 命令将指定库中的一个或多个符号导出到 SVG 文件中。

用法：`kicad-cli sym export svg [--help] [--output OUTPUT_DIR] [--theme THEME_NAME] [--symbol SYMBOL] [--black-and-white] [--include-hidden-pins] [--include-hidden-fields] INPUT_FILE`

位置参数：

INPUT_FILE	用于导出的符号库文件。
------------	-------------

可选参数：

<code>-h, --help</code>	Show help for the symbol SVG export command.
<code>-o &lt;output dir&gt;, --output &lt;output dir&gt;</code>	The output folder for the exported files. Each symbol in the input library is output to a separate file. When <code>--output</code> is not used, the files are exported to the current directory.
<code>-t &lt;theme name&gt;, --theme &lt;theme name&gt;</code>	The name of the theme to use for export. If no theme is given, the symbol editor's currently selected theme is used.
<code>-s &lt;symbol name&gt;, --symbol &lt;symbol name&gt;</code>	The specific symbol to export from the library. When this argument is not used, all symbols in the library are exported.
<code>--black-and-white</code>	Export symbols in black and white.
<code>--include-hidden-pins</code>	Export hidden pins in the exported SVG.
<code>--include-hidden-fields</code>	Export hidden symbol fields in the exported SVG.

## 符号升级

The `sym upgrade` command converts the specified symbol library from a legacy KiCad symbol format or a non-KiCad symbol format to the native format for the current version of KiCad. If the input library is already in the current file format, no action is taken.

Supported input symbol formats are:

- KiCad symbol library ( `.kicad_sym` )
- KiCad (pre-6.0) symbol library ( `.lib` )
- Altium schematic library ( `.SchLib` )
- Altium integrated library ( `.IntLib` )
- CADSTAR parts library ( `.lib` )

EAGLE XML library ( `.lbr` )

- EasyEDA (JLCEDA) Std file ( `.json` )
- EasyEDA (JLCEDA) Pro file ( `.elibz` , `.epro` , `.zip` )

用法： `kiCad-cli sym upgrade [--help] [--output OUTPUT_FILE] [--force] INPUT_FILE`

位置参数：

INPUT_FILE	要升级的符号库。
------------	----------

可选参数：

<code>-h</code> , <code>--help</code>	Show help for the symbol upgrade command.
<code>-o &lt;output filename&gt;</code> , <code>--output &lt;output filename&gt;</code>	The output filename for the upgraded symbol library. When <code>--output</code> is not used, the upgraded symbol library is saved over the original library.
<code>--force</code>	Re-save the input library even if it is already in the current file format.

# 版本命令

The `version` command prints the KiCad version. Without any arguments, it simply prints the version number, for example `7.0.7`. You can print the version in several other formats using the `--format` argument.

NOTE

在 Gitlab 上提交错误报告或功能请求时，请使用 `kicad-cli version --format about` 获取要包含的版本信息。

用法： `kicad-cli version [--help] [--format VAR]`

可选参数：

<code>--format &lt;format&gt;</code>	版本号的格式。选项为 <code>plain</code> （默认）、 <code>commit</code> 或 <code>about</code> 。 <code>plain</code> 打印版本号（例如 <code>7.0.7</code> ），如果不使用 <code>--format</code> 参数，则为默认值。 <code>commit</code> 打印您正在使用的 KiCad 构建的 git commit 的哈希值。 <code>about</code> 打印完整版本信息，包括库版本和基本系统信息。您可以在 Bug 报告中 <a href="#">使用关于</a> 版本信息。
--------------------------------------	--