# KiCad 命令行界面

The KiCad Team

# Table of Contents

*KiCad 10.0 参考手册*

**Copyright**



**Contributors**

Graham Keeth

**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**

本用户手册基于 KiCad 10.0.0 版本。其他版本的 KiCad 在功能和界面外观上可能存在差异。

文档修订版：`90e4b228`。

# 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`。

`kicad-cli` 命令包含 6 个子命令：`fp`、`jobset`、`pcb`、`sch`、`sym` 和 `version`。每个子命令都可以有自己的子命令和参数。例如，要从 PCB 导出 Gerber 文件，您可以运行 `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_FILE_OR_DIR`

位置参数：

| | |
|---|---|
| `INPUT_FILE_OR_DIR` | Footprint ( `.kicad_mod` ) or footprint library directory ( `.pretty` ) to export. |

可选参数：

| | |
|---|---|
| `-h`, `--help` | Show help for the footprint SVG export command. |
| `-o <output dir>`, `--output <output dir>` | The output folder for the exported files. One file is output for each layer of each footprint in the library. 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 export from the footprint, such as `F.Cu,B.Cu`. If no layers are given, all layers are exported. Layer names can be specified as canonical layer names (`F.Cu`, `In.1`, `F.Fab`, etc.) or as user-defined (custom) layer names, but user-defined layer names are matched first. |
| `-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 footprint editor's currently selected theme is used. |
| `--fp <footprint>`, `--footprint <footprint>` | The name of the specific footprint to export from the library. When this argument is not used, all footprints in the library are exported. |
| `--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. |
| `--cdnp`, `--crossout-DNP-footprints-on-fab-layers` | Plot an "X" over the courtyard of DNP footprints on fab layers, and strikeout their reference designators. |
| `--black-and-white` | 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.

支持的输入封装格式为:

- KiCad 封装库（包含 `.kicad_mod` 文件的 `.pretty` 文件夹）
- KiCad（5.0 之前版本）封装库（`.mod`、`.emp`）
- Altium 封装库（`.PcbLib`）
- Altium 集成库（`.IntLib`）
- CADSTAR PCB 存档文件（`.cpa`）
- EAGLE XML 库（`.lbr`）

- EasyEDA (JLCEDA) 标准版文件（`.json`）
- EasyEDA (JLCEDA) 专业版文件（`.elibz`, `.epro`, `.zip`）
- GEDA/PCB 库（包含 `.fp` 文件的文件夹）

Usage: `kicad-cli fp upgrade [--help] [--output OUTPUT_DIR] [--force] INPUT_FILE_OR_DIR`

位置参数：

| | |
|---|---|
| INPUT_FILE_OR_DIR | Footprint or footprint library directory to upgrade. For KiCad format footprint libraries, this can be a footprint (`.kicad_mod` file) or a footprint library (`.pretty` directory containing `.kicad_mod` files). |

可选参数：

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

# jobset 命令

`jobset run` 命令用于运行预先定义的 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：

| | |
|---|---|
| `-h`, `--help` | Show help for the jobset command. |
| `--stop-on-error` | As jobs are executed in sequence, stop running after a job fails. If not given, jobs will continue executing after any job fails. |
| `-f <jobset file>`, `--file <jobset file>` | The jobset file (`.kicad_jobset`) to run. |
| `--output <destination description or ID>` | The jobset destination to generate. If no destination is specified, all destinations will be generated.<br><br>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.<br><br>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 `jobset run` command when `--output` is not used. It can also be found in the `.kicad_jobset` file under the destination's `id` key. |

# PCB 命令

`pcb` 命令用于执行设计规则检查或将电路板导出为多种其他文件格式，包括制造文件和 3D 文件。

## PCB DRC

`pcb drc` 命令对电路板执行设计规则检查并生成报告。

Usage: `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] [--refill-zones] [--save-board] INPUT_FILE`

位置参数：

| | |
|---|---|
| `INPUT_FILE` | Board file to run DRC on. |

可选参数：

| | |
|---|---|
| -h, --help | Show help for the DRC command. |
| -o <output filename>, --output <output filename> | Output filename for the generated DRC report. When --output is not used, the output filename will be the same as the input file, with the .rpt or .json file extension, depending on the selected format. |
| -D <variable name>=<value>, --define-var <variable_name>=<value> | Add or override project variable definitions. Can be used multiple times to define multiple variables. |
| --format <format> | Report file format. Options are report (default) or json. |
| --all-track-errors | Report all errors for each track. |
| --schematic-parity | Test for parity between PCB and schematic. |
| --units <unit> | Units to use in the report. Options are mm (default), in, or mils. |
| --severity-all | Report all DRC violations. This is equivalent to using all of the other DRC severity options. |
| --severity-error | Report all error-level DRC violations. This can be combined with the other DRC severity options. |
| --severity-warning | Report all warning-level DRC violations. This can be combined with the other DRC severity options. |
| --severity-exclusions | Report all excluded DRC violations. This can be combined with the other DRC severity options. |
| --exit-code-violations | 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. |
| --refill-zones | Refill all zones before running DRC. The board will not be saved after refilling zones unless --save-board is also used. |
| --save-board | Save the board after running DRC. The board will not be saved unless --refill-zones is also used. |

## PCB export: 3D PDF

The `pcb export 3dpdf` command exports a board design to a PDF file containing an embedded 3D model of the board.

Usage: `kicad-cli pcb export 3dpdf [--help] [--output OUTPUT_FILE] [--define-var KEY=VALUE]… [--force] [--no-unspecified] [--no-dnp] [--variant VAR]… [--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] [--no-extra-pad-thickness] [--min-distance MIN_DIST] [--net-filter VAR] [--user-origin VAR] INPUT_FILE`

位置参数：

| | |
|---|---|
| INPUT_FILE | Board file to export. |

可选参数：

| | |
|---|---|
| -h , --help | Show help for the 3D PDF 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 .pdf 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. |
| --variant <variant name> | The name(s) of the variant(s) to output. Can be used multiple times to output multiple variants. When specifying multiple variants, use ${VARIANT} in the output path to generate separate files for each variant. When --variant is not used, the default variant is output. |
| --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. |

| | |
|---|---|
| `--include-soldermask` | Include solder mask layers in export as a set of flat faces. |
| `--fuse-shapes` | Fuse overlapping geometry together in export (time consuming). |
| `--fill-all-vias` | Don’t cut via holes in conductor layers. |
| `--no-extra-pad-thickness` | Disable adding additional metal thickness to pads. When not used, pads have 0.005mm added to their metal thickness, which causes pads to be separate faces in the exported model, distinct from the surrounding metal. |
| `--min-distance <min distance>` | Tolerance for considering two points to be in the same location. Default: `0.01mm`. |
| `--net-filter <net filter>` | Only include copper items belonging to nets matching this wildcard. |
| `--user-origin <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 export: BREP (OCCT)

`pcb export brep` 命令将电路板设计导出为 BREP（OCCT 原生边界表示）3D 模型文件。

Usage: `kicad-cli pcb export brep [--help] [--output OUTPUT_FILE] [--define-var KEY=VALUE]… [--force] [--no-unspecified] [--no-dnp] [--variant VAR]… [--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] [--no-extra-pad-thickness] [--min-distance MIN_DIST] [--net-filter VAR] [--user-origin VAR] INPUT_FILE`

位置参数：

| | |
|---|---|
| `INPUT_FILE` | Board file to export. |

可选参数：

| | |
|---|---|
| `-h`, `--help` | Show help for the BREP 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 `.brep` 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. |
| `--variant <variant name>` | The name(s) of the variant(s) to output. Can be used multiple times to output multiple variants. When specifying multiple variants, use `${VARIANT}` in the output path to generate separate files for each variant. When `--variant` is not used, the default variant is output. |
| `--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. |
| `--include-soldermask` | Include solder mask layers in export as a set of flat faces. |
| `--fuse-shapes` | Fuse overlapping geometry together in export (time consuming). |

| | |
|---|---|
| `--fill-all-vias` | Don't cut via holes in conductor layers. |
| `--no-extra-pad-thickness` | Disable adding additional metal thickness to pads. When not used, pads have 0.005mm added to their metal thickness, which causes pads to be separate faces in the exported model, distinct from the surrounding metal. |
| `--min-distance <min distance>` | Tolerance for considering two points to be in the same location. Default: `0.01mm`. |
| `--net-filter <net filter>` | Only include copper items belonging to nets matching this wildcard. |
| `--user-origin <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 export: drill file

`pcb export drill` 命令从电路板导出钻孔文件。

Usage: `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] [--generate-report] [--report-path REPORT_FILE] [--generate-tenting] [--map-format MAP_FORMAT] [--gerber-precision VAR] INPUT_FILE`

位置参数：

| | |
|---|---|
| `INPUT_FILE` | Board file to export. |

可选参数：

| | |
|---|---|
| `-h`, `--help` | Show help for the drill file export command. |
| `-o <output dir>`, `--output <output dir>` | The output directory for the drill file(s). When `--output` is not used, the drill file(s) are saved in the current directory. |
| `--format <format>` | The drill file format. Options are `excellon` (default) or `gerber`. |
| `--drill-origin <origin>` | The coordinate origin for the drill file. Options are `absolute` (default) to use the board's absolute origin or `plot` to use the board's drill/placement origin. |
| `--excellon-zeros-format <format>` | The zeros format for the drill file. Options are `decimal` (default), `suppressleading`, `suppresstrailing`, or `keep`. Only applies to Excellon format drill files. |
| `--excellon-oval-format <format>` | Control the oval holes drill mode. Options are `route` and `alternate` (default). Only applies to Excellon format drill files. |
| `-u <units>`, `--excellon-units <units>` | The units for the drill file. Options are `mm` (default) or `in`. Only applies to Excellon format drill files. |
| `--excellon-mirror-y` | Mirror the drill file in the Y direction. Only applies to Excellon format drill files. |
| `--excellon-min-header` | Use a minimal header in the drill file. Only applies to Excellon format drill files. |
| `--excellon-separate-th` | Generate separate drill files for plated and non-plated through holes. Only applies to Excellon format drill files. |
| `--generate-map` | Generate a map file in addition to the drill file. |
| `--generate-report` | Generate a report file listing all drill hits. |
| `--report-path <report filename>` | The output filename for the drill report file. When `--report-path` is not used, the report filename will be the same as the input file, with the `-drill.rpt` suffix and file extension. |
| `--generate-tenting` | Generate separate drill files for tented drill hits. Only applies to Gerber X2 format drill files. |
| `--map-format <format>` | The map file format. Options are `pdf` (default), `gerberx2`, `ps`, `dxf`, or `svg`. |
| `--gerber-precision <precision>` | The precision (number of digits) for the drill file. Valid options are `5` or `6` (default). Only applies to Gerber format drill files. |

## PCB export: DXF

`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_DIR] [--layers LAYER_LIST] [--common-layers COMMON_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] [--mode-single] [--mode-multi] [--scale SCALE] [--check-zones] [--variant VAR]… INPUT_FILE

位置参数：

| | |
|---|---|
| INPUT_FILE | Board file to export. |

可选参数：

| | |
|---|---|
| -h , --help | Show help for the DXF 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 footprint, 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 user-defined layer names are matched first. |
| --cl <layer list> , --common-layers <layer list> | 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 user-defined 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. |
| --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. |

| | |
|---|---|
| --cdnp , --crossout-DNP-footprints-on-fab-layers | Plot an "X" over the courtyard of DNP footprints on fab layers, and strikeout their reference designators. |
| --subtract-soldermask | Remove silkscreen from areas without soldermask. |
| --uc , --use-contours | Plot graphic items using their contours. |
| --udo , --use-drill-origin | Plot using the drill/place file origin. |
| --ibt , --include-border-title | Include sheet border and title block in plot. |
| --ou <unit> , --output-units <unit> | Output units. Options are mm or in (default). |
| --drill-shape-opt <shape> | 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). |
| --mode-single | 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. |
| --mode-multi | Plot the layers to one or more DXF files, with each file representing a single layer from LAYER_LIST. The output path specifies the directory in which the files will be written. |
| --scale <scale> | A scaling factor to use for plotting the PCB. The border and title block are not scaled. A scale factor of 0 autoscales the plot. |
| --check-zones | Check zone fills and refill zones, if required, prior to export. Any zone fill updates are not saved in the board file. |
| --variant <variant name> | The name(s) of the variant(s) to output. Can be used multiple times to output multiple variants. When specifying multiple variants, use ${VARIANT} in the output path to generate separate files for each variant. When --variant is not used, the default variant is output. |

## PCB export: GenCAD

`pcb export gencad` 命令将电路板设计导出为 GenCAD 文件。

Usage: `kicad-cli pcb export gencad [--help] [--output OUTPUT_DIR] [--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 | Board file to export. |

可选参数：

| | |
|---|---|
| `-h`, `--help` | Show help for the DXF 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 `.cad` 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`, `--flip-bottom-pads` | Flip bottom footprint padstacks. |
| `--unique-pins` | Generate unique pin names. |
| `--unique-footprints` | Generate a new shape for each footprint instance (do not reuse shapes). |
| `--use-drill-origin` | Use drill/place file origin as origin. |
| `--store-origin-coord` | Save the origin coordinates in the file. |

## PCB export: Gerber

`pcb export gerbers` 命令将电路板设计导出为 Gerber 文件，每个文件对应一层。

Usage: `kicad-cli pcb export gerbers [--help] [--output OUTPUT_DIR] [--layers LAYER_LIST] [--common-layers COMMON_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] [--check-zones] [--variant VAR]… [--board-plot-params] INPUT_FILE`

位置参数：

| | |
|---|---|
| INPUT_FILE | Board file to export. |

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 user-defined layer names are matched first. |
| `--cl <layer list>`, `--common-layers <layer list>` | A comma-separated list of layer names to plot on all layers, such as `F.Cu,B.Cu`. Each layer specified is included in every output file. Layer names can be specified as canonical layer names ( `F.Cu`, `In.1`, `F.Fab`, etc.) or as user-defined (custom) layer names, but user-defined 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. |
| `--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. |
| `--cdnp`, `--crossout-DNP-footprints-on-fab-layers` | Plot an "X" over the courtyard of DNP footprints on fab layers, and strikeout their reference designators. |
| `--no-x2` | Do not use the extended X2 format. |
| `--no-netlist` | Do not include netlist attributes. |

| | |
|---|---|
| `--subtract-soldermask` | Remove silkscreen from areas without soldermask. |
| `--disable-aperture-macros` | Disable aperture macros. |
| `--use-drill-file-origin` | Use drill/place file origin instead of absolute origin. |
| `--precision <precision>` | The precision (number of digits) for the Gerber files. Valid options are `5` or `6` (default). |
| `--no-protel-ext` | Use `.gbr` file extension instead of Protel file extensions ( `.gbl` , `.gtl` , etc.). |
| `--check-zones` | Check zone fills and refill zones, if required, prior to export. Any zone fill updates are not saved in the board file. |
| `--variant <variant name>` | The name(s) of the variant(s) to output. Can be used multiple times to output multiple variants. When specifying multiple variants, use `${VARIANT}` in the output path to generate separate files for each variant. When `--variant` is not used, the default variant is output. |
| `--board-plot-params` | Use the Gerber plot settings already configured in the board file. |

## PCB export: GLB

`pcb export glb` 命令将电路板设计导出为 GLB（二进制 glTF）3D 模型文件。

Usage: `kicad-cli pcb export glb [--help] [--output OUTPUT_FILE] [--define-var KEY=VALUE]… [--force] [--no-unspecified] [--no-dnp] [--variant VAR]… [--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] [--no-extra-pad-thickness] [--min-distance MIN_DIST] [--net-filter VAR] [--user-origin VAR] INPUT_FILE`

位置参数：

| | |
|---|---|
| `INPUT_FILE` | Board file to export. |

可选参数：

| | |
|---|---|
| `-h` , `--help` | Show help for the GLB 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 `.glb` 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. |
| --variant <variant name> | The name(s) of the variant(s) to output. Can be used multiple times to output multiple variants. When specifying multiple variants, use ${VARIANT} in the output path to generate separate files for each variant. When --variant is not used, the default variant is output. |
| --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. |
| --include-soldermask | Include solder mask layers in export as a set of flat faces. |
| --fuse-shapes | Fuse overlapping geometry together in export (time consuming). |

| | |
|---|---|
| `--fill-all-vias` | Don't cut via holes in conductor layers. |
| `--no-extra-pad-thickness` | Disable adding additional metal thickness to pads. When not used, pads have 0.005mm added to their metal thickness, which causes pads to be separate faces in the exported model, distinct from the surrounding metal. |
| `--min-distance <min distance>` | Tolerance for considering two points to be in the same location. Default: `0.01mm`. |
| `--net-filter <net filter>` | Only include copper items belonging to nets matching this wildcard. |
| `--user-origin <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 export: HPGL

**NOTE** `kicad-cli pcb export hpgl` is not functional in KiCad 10.0.

The `pcb export hpgl` command is not functional in KiCad 10.0 as KiCad no longer supports HPGL output. In previous versions of KiCad it exported a board design to an HPGL file. It is included as a non-functional command for compatibility reasons. It will be removed in a future version of KiCad.

Usage: `kicad-cli pcb export hpgl [--help] [--output OUTPUT_DIR] INPUT_FILE`

## PCB export: IPC-2581

`pcb export ipc2581` 命令将电路板设计导出为 IPC-2581 格式。

Usage: `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] [--variant VAR]… INPUT_FILE`

位置参数：

| | |
|---|---|
| `INPUT_FILE` | Board file to export. |

可选参数：

| | |
|---|---|
| `-h`, `--help` | Show help for the IPC-2581 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 `.xml` file extension. |
| `--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. |
| `--precision <precision>` | The precision (number of digits after the decimal separator) for the exported file. The default is `6`. |
| `--compress` | Compress output file as a ZIP file. |
| `--version <IPC-2581 standard version>` | IPC-2581 standard version to use. Options are `B` or `C` (default). |
| `--units` | Units to use in export. Options are `mm` (default) or `in`. |
| `--bom-col-int-id` | 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. |
| `--bom-col-mfg-pn` | 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. |
| `--bom-col-mfg` | 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. |
| `--bom-col-dist-pn` | 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. |
| `--bom-col-dist` | 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. |
| `--bom-rev` | Revision to use for the Bill of Materials Revision field. If not given, the `Revision` field from the schematic's root sheet is used instead. |
| `--variant <variant name>` | The name(s) of the variant(s) to output. Can be used multiple times to output multiple variants. When specifying multiple variants, use `${VARIANT}` in the output path to generate separate files for each variant. When `--variant` is not used, the default variant is output. |

## PCB export: IPC-D-356

`pcb export ipcd356` 命令从电路板设计中生成 IPC-D-356 网表。

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

位置参数：

| | |
|---|---|
| `INPUT_FILE` | Board file to export. |

可选参数：

| | |
|---|---|
| `-h` , `--help` | Show help for the IPC-D-356 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 `.d356` file extension. |

## PCB export: ODB++

`pcb export odb` 命令将电路板设计导出为 ODB++ 格式。

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

位置参数：

| | |
|---|---|
| `INPUT_FILE` | Board file to export. |

可选参数：

| | |
|---|---|
| `-h` , `--help` | Show help for the ODB++ export command. |
| `-o <output filename>` , `--output <output filename>` | The output filename, or folder name if no compression is used. |
| `--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. |
| `--precision <precision>` | The precision (number of digits after the decimal separator) for the exported file. The default is 2. |
| `--compression <mode>` | Compression mode. Options are `none` , `zip` (default), or `tgz` . |
| `--units <unit>` | Units to use in the output file. Options are `mm` (default) or `in` . |
| `--variant <variant name>` | The name(s) of the variant(s) to output. Can be used multiple times to output multiple variants. When specifying multiple variants, use `${VARIANT}` in the output path to generate separate files for each variant. When `--variant` is not used, the default variant is output. |

## PCB export: PDF

`pcb export pdf` 命令将电路板设计导出为 PDF 文件。每个层可以单独导出为独立的文件，也可以作为单个文件中的一个页面进行导出。

Usage: `kicad-cli pcb export pdf [--help] [--output OUTPUT_DIR] [--layers LAYER_LIST] [--common-layers COMMON_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] [--mode-single] [--mode-separate] [--mode-multipage] [--scale SCALE] [--bg-color COLOR] [--check-zones] [--variant VAR]… INPUT_FILE`

位置参数：

| | |
|---|---|
| INPUT_FILE | Board file to export. |

可选参数：

| | |
|---|---|
| `-h`, `--help` | Show help for the PDF export command. |
| -o <output dir>, --output <output dir> | The output folder or filename for the exported files. When `--mode-single` or `--mode-multipage` 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 `.pdf` file extension. When `--mode-separate` 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 user-defined layer names are matched first. |
| --cl <layer list>, --common-layers <layer list> | 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 user-defined 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. |
| `-m`, `--mirror` | Mirror the board. This can be useful for showing bottom layers. |
| `--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. |
| `--subtract-soldermask` | Remove silkscreen from areas without soldermask. |

| | |
|---|---|
| `--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. |
| `--cdnp` , `--crossout-DNP-footprints-on-fab-layers` | Plot an "X" over the courtyard of DNP footprints on fab layers, and strikeout their reference designators. |
| `-n` , `--negative` | Plot in negative. |
| `--black-and-white` | Plot in black and white. |
| `-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. |
| `--drill-shape-opt` | 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). |
| `--mode-single` | 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. All specified layers are plotted on a single page. |
| `--mode-separate` | Plot the layers to one or more PDF files, with each file representing a single layer from `LAYER_LIST`. The output path specifies the directory in which the files will be written. |
| `--mode-multipage` | Plot the layers to a single PDF file with multiple pages, with each page representing a single layer from `LAYER_LIST`. The output path specifies the complete directory and filename path. |
| `--scale <scale>` | A scaling factor to use for plotting the PCB. The border and title block are not scaled. A scale factor of 0 autoscales the plot. |
| `--bg-color <color>` | A background color for the plot. The format can be hex ( `#rrggbb` or `#rrggbbaa` ) or CSS ( `rgb(r,g,b)` or `rgba(r,g,b,a)` ). |
| `--check-zones` | Check zone fills and refill zones, if required, prior to export. Any zone fill updates are not saved in the board file. |

| | |
|---|---|
| `--variant <variant name>` | The name(s) of the variant(s) to output. Can be used multiple times to output multiple variants. When specifying multiple variants, use `${VARIANT}` in the output path to generate separate files for each variant. When `--variant` is not used, the default variant is output. |

## PCB export: PLY

`pcb export ply` 命令将电路板设计导出为 PLY 3D 模型文件。

Usage: `kicad-cli pcb export ply [--help] [--output OUTPUT_FILE] [--define-var KEY=VALUE]… [--force] [--no-unspecified] [--no-dnp] [--variant VAR]… [--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] [--no-extra-pad-thickness] [--min-distance MIN_DIST] [--net-filter VAR] [--user-origin VAR] INPUT_FILE`

位置参数：

| | |
|---|---|
| `INPUT_FILE` | Board file to export. |

可选参数：

| | |
|---|---|
| `-h`, `--help` | Show help for the PLY 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 `.ply` 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. |
| `--variant <variant name>` | The name(s) of the variant(s) to output. Can be used multiple times to output multiple variants. When specifying multiple variants, use `${VARIANT}` in the output path to generate separate files for each variant. When `--variant` is not used, the default variant is output. |
| `--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. |
| `--include-soldermask` | Include solder mask layers in export as a set of flat faces. |
| `--fuse-shapes` | Fuse overlapping geometry together in export (time consuming). |
| `--fill-all-vias` | Don't cut via holes in conductor layers. |
| `--no-extra-pad-thickness` | Disable adding additional metal thickness to pads. When not used, pads have 0.005mm added to their metal thickness, which causes pads to be separate faces in the exported model, distinct from the surrounding metal. |
| `--min-distance <min distance>` | Tolerance for considering two points to be in the same location. Default: `0.01mm`. |
| `--net-filter <net filter>` | Only include copper items belonging to nets matching this wildcard. |
| `--user-origin <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 export: position file

`pcb export pos` 命令从电路板设计中导出位置文件。

Usage: `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] [--variant VAR]… INPUT_FILE`

位置参数：

| | |
|---|---|
| INPUT_FILE | Board file to export. |

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). |
| `--variant <variant name>` | The name(s) of the variant(s) to output. Can be used multiple times to output multiple variants. When specifying multiple variants, use `${VARIANT}` in the output path to generate separate files for each variant. When `--variant` is not used, the default variant is output. |

## PCB export: PostScript

`pcb export ps` 命令将电路板设计导出为 PostScript 文件。

Usage: `kicad-cli pcb export ps [--help] [--output OUTPUT_DIR] [--layers LAYER_LIST] [--common-layers COMMON_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] [--mode-single] [--mode-multi] [--track-width-correction TRACK_COR] [--x-scale-factor X_SCALE] [--y-scale-factor Y_SCALE] [--force-a4] [--scale SCALE] [--check-zones] [--variant VAR]… INPUT_FILE`

位置参数：

| | |
|---|---|
| `INPUT_FILE` | Board file to export. |

可选参数：

| | |
|---|---|
| `-h`, `--help` | Show help for the PS 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 `.ps` 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 user-defined layer names are matched first. |
| `--cl <layer list>`, `--common-layers <layer list>` | A comma-separated list of layer names to plot on all layers, such as `F.Cu,B.Cu`. Each layer specified is included in every output file. Layer names can be specified as canonical layer names (`F.Cu`, `In.1`, `F.Fab`, etc.) or as user-defined (custom) layer names, but user-defined 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. |
| `-m`, `--mirror` | Mirror the board. This can be useful for showing bottom layers. |
| `--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. |
| `--subtract-soldermask` | Remove silkscreen from areas without soldermask. |
| `--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. |

| | |
|---|---|
| `--cdnp`, `--crossout-DNP-footprints-on-fab-layers` | Plot an "X" over the courtyard of DNP footprints on fab layers, and strikeout their reference designators. |
| `-n`, `--negative` | Plot in negative. |
| `--black-and-white` | Plot in black and white. |
| `-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. |
| `--drill-shape-opt` | 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). |
| `--mode-single` | 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. |
| `--mode-multi` | Plot the layers to one or more PS files, with each file representing a single layer from `LAYER_LIST`. The output path specifies the directory in which the files will be written. |
| `-C`, `--track-width-correction` | A global correction, in millimeters, that is added to the size of tracks, vias, and pads when plotted. This correction can be used to correct for errors in the PostScript output device to achieve an exact-scale output. |
| `-X`, `--x-scale-factor` | X scale adjust for exact scale. |
| `-Y`, `--y-scale-factor` | Y scale adjust for exact scale. |
| `-A`, `--force-a4` | Force A4 paper size. |
| `--scale <scale>` | A scaling factor to use for plotting the PCB. The border and title block are not scaled. A scale factor of 0 autoscales the plot. |
| `--check-zones` | Check zone fills and refill zones, if required, prior to export. Any zone fill updates are not saved in the board file. |
| `--variant <variant name>` | The name(s) of the variant(s) to output. Can be used multiple times to output multiple variants. When specifying multiple variants, use `${VARIANT}` in the output path to generate separate files for each variant. When `--variant` is not used, the default variant is output. |

## PCB export: statistics

The `pcb export stats` command exports a report of statistics about the board design.

Usage: `kicad-cli pcb export stats [--help] [--output OUTPUT_FILE] [--format FORMAT] [--units UNITS] [--exclude-footprints-without-pads] [--subtract-holes-from-board] [--subtract-holes-from-copper] INPUT_FILE`

位置参数：

| | |
|---|---|
| `INPUT_FILE` | Board file to export statistics from. |

可选参数：

| | |
|---|---|
| `-h`, `--help` | Show help for the statistics command. |
| `-o <output filename>`, `--output <output filename>` | Output filename for the generated statistics report. When `--output` is not used, the output filename will be the same as the input file, with a `_statistics` suffix and the `.rpt` or `.json` file extension, depending on the selected format. |
| `--format <format>` | Report file format. Options are `report` (default) or `json`. |
| `--units <unit>` | Units to use in the report. Options are `mm` (default) or `in`. |
| `--exclude-footprints-without-pads` | Exclude footprints that do not contain any pads from component counts. |
| `--subtract-holes-from-board` | Subtract the area of holes from the total board area. |
| `--subtract-holes-from-copper` | Subtract the area of holes from the total copper area. |

## PCB export: STEP

`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] [--variant VAR]… [--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] [--no-extra-pad-thickness] [--min-distance MIN_DIST] [--net-filter VAR] [--no-optimize-step] [--user-origin VAR] INPUT_FILE`

位置参数：

| | |
|---|---|
| `INPUT_FILE` | Board file to export. |

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. |
| --variant <variant name> | The name(s) of the variant(s) to output. Can be used multiple times to output multiple variants. When specifying multiple variants, use ${VARIANT} in the output path to generate separate files for each variant. When --variant is not used, the default variant is output. |
| --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. |

| | |
|---|---|
| `--include-soldermask` | Include solder mask layers in export as a set of flat faces. |
| `--fuse-shapes` | Fuse overlapping geometry together in export (time consuming). |
| `--fill-all-vias` | Don't cut via holes in conductor layers. |
| `--no-extra-pad-thickness` | Disable adding additional metal thickness to pads. When not used, pads have 0.005mm added to their metal thickness, which causes pads to be separate faces in the exported model, distinct from the surrounding metal. |
| `--min-distance <min distance>` | Tolerance for considering two points to be in the same location. Default: `0.01mm`. |
| `--net-filter <net filter>` | Only include copper items belonging to nets matching this wildcard. |
| `--no-optimize-step` | 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. |
| `--user-origin <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 export: STL

`pcb export stl` 命令将电路板设计导出为 STL 3D 模型文件。

Usage: `kicad-cli pcb export stl [--help] [--output OUTPUT_FILE] [--define-var KEY=VALUE]… [--force] [--no-unspecified] [--no-dnp] [--variant VAR]… [--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] [--no-extra-pad-thickness] [--min-distance MIN_DIST] [--net-filter VAR] [--user-origin VAR] INPUT_FILE`

位置参数：

| | |
|---|---|
| `INPUT_FILE` | Board file to export. |

可选参数：

| | |
|---|---|
| `-h`, `--help` | Show help for the STL 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 .stl 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. |
| --variant <variant name> | The name(s) of the variant(s) to output. Can be used multiple times to output multiple variants. When specifying multiple variants, use ${VARIANT} in the output path to generate separate files for each variant. When --variant is not used, the default variant is output. |
| --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. |

| | |
|---|---|
| `--include-soldermask` | Include solder mask layers in export as a set of flat faces. |
| `--fuse-shapes` | Fuse overlapping geometry together in export (time consuming). |
| `--fill-all-vias` | Don't cut via holes in conductor layers. |
| `--no-extra-pad-thickness` | Disable adding additional metal thickness to pads. When not used, pads have 0.005mm added to their metal thickness, which causes pads to be separate faces in the exported model, distinct from the surrounding metal. |
| `--min-distance <min distance>` | Tolerance for considering two points to be in the same location. Default: `0.01mm`. |
| `--net-filter <net filter>` | Only include copper items belonging to nets matching this wildcard. |
| `--user-origin <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 export: STEPZ

The `pcb export stpz` command exports a board design to a STEPZ (GZIP-compressed STEP) file.

Usage: `kicad-cli pcb export stpz [--help] [--output OUTPUT_FILE] [--define-var KEY=VALUE]… [--force] [--no-unspecified] [--no-dnp] [--variant VAR]… [--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] [--no-extra-pad-thickness] [--min-distance MIN_DIST] [--net-filter VAR] [--no-optimize-step] [--user-origin VAR] INPUT_FILE`

位置参数：

| | |
|---|---|
| `INPUT_FILE` | Board file to export. |

可选参数：

| | |
|---|---|
| `-h`, `--help` | Show help for the STEPZ 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 `.stpz` 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. |
| `--variant <variant name>` | The name(s) of the variant(s) to output. Can be used multiple times to output multiple variants. When specifying multiple variants, use `${VARIANT}` in the output path to generate separate files for each variant. When `--variant` is not used, the default variant is output. |
| `--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. |
| `--include-soldermask` | Include solder mask layers in export as a set of flat faces. |
| `--fuse-shapes` | Fuse overlapping geometry together in export (time consuming). |

| | |
|---|---|
| `--fill-all-vias` | Don't cut via holes in conductor layers. |
| `--no-extra-pad-thickness` | Disable adding additional metal thickness to pads. When not used, pads have 0.005mm added to their metal thickness, which causes pads to be separate faces in the exported model, distinct from the surrounding metal. |
| `--min-distance <min distance>` | Tolerance for considering two points to be in the same location. Default: `0.01mm`. |
| `--net-filter <net filter>` | Only include copper items belonging to nets matching this wildcard. |
| `--no-optimize-step` | 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. |
| `--user-origin <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 export: U3D

The `pcb export u3d` command exports a board design to a PDF file containing an embedded 3D model of the board.

Usage: `kicad-cli pcb export u3d [--help] [--output OUTPUT_FILE] [--define-var KEY=VALUE]… [--force] [--no-unspecified] [--no-dnp] [--variant VAR]… [--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] [--no-extra-pad-thickness] [--min-distance MIN_DIST] [--net-filter VAR] [--user-origin VAR] INPUT_FILE`

位置参数：

| | |
|---|---|
| `INPUT_FILE` | Board file to export. |

可选参数：

| | |
|---|---|
| `-h`, `--help` | Show help for the 3D PDF 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 `.pdf` 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. |
| --variant <variant name> | The name(s) of the variant(s) to output. Can be used multiple times to output multiple variants. When specifying multiple variants, use ${VARIANT} in the output path to generate separate files for each variant. When --variant is not used, the default variant is output. |
| --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. |
| --include-soldermask | Include solder mask layers in export as a set of flat faces. |
| --fuse-shapes | Fuse overlapping geometry together in export (time consuming). |

| | |
|---|---|
| `--fill-all-vias` | Don't cut via holes in conductor layers. |
| `--no-extra-pad-thickness` | Disable adding additional metal thickness to pads. When not used, pads have 0.005mm added to their metal thickness, which causes pads to be separate faces in the exported model, distinct from the surrounding metal. |
| `--min-distance <min distance>` | Tolerance for considering two points to be in the same location. Default: `0.01mm`. |
| `--net-filter <net filter>` | Only include copper items belonging to nets matching this wildcard. |
| `--user-origin <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 export: SVG

`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_DIR] [--layers LAYER_LIST] [--common-layers COMMON_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] [--mode-single] [--mode-multi] [--scale SCALE] [--check-zones] [--variant VAR]… INPUT_FILE`

位置参数：

| | |
|---|---|
| `INPUT_FILE` | Board file to export. |

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 user-defined layer names are matched first. |
| `--cl <layer list>`, `--common-layers <layer list>` | 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 user-defined 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. |
| `--cdnp`, `--crossout-DNP-footprints-on-fab-layers` | Plot an "X" over the courtyard of DNP footprints on fab layers, and strikeout their reference designators. |

| | |
|---|---|
| --page-size-mode <mode> | 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. |
| --fit-page-to-board | Set the SVG size to match the board outline. This is equivalent to --page-size-mode 2. |
| --exclude-drawing-sheet | Plot SVG without a drawing sheet. |
| --drill-shape-opt | 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). |
| --mode-single | 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. |
| --mode-multi | 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. |
| --scale <scale> | A scaling factor to use for plotting the PCB. The border and title block are not scaled. A scale factor of 0 autoscales the plot. |
| --check-zones | Check zone fills and refill zones, if required, prior to export. Any zone fill updates are not saved in the board file. |
| --variant <variant name> | The name(s) of the variant(s) to output. Can be used multiple times to output multiple variants. When specifying multiple variants, use ${VARIANT} in the output path to generate separate files for each variant. When --variant is not used, the default variant is output. |

## PCB export: VRML

pcb export vrml 命令将电路板设计导出为 VRML 3D 文件。

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

位置参数：

| | |
|---|---|
| INPUT_FILE | Board file to export. |

可选参数：

| | |
|---|---|
| `-h`, `--help` | Show help for the VRML 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 `.wrl` 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. |
| `--variant <variant name>` | The name(s) of the variant(s) to output. Can be used multiple times to output multiple variants. When specifying multiple variants, use `${VARIANT}` in the output path to generate separate files for each variant. When `--variant` is not used, the default variant is output. |
| `--user-origin <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. If this option is not given, the board center is used. |
| `--units <units>` | Units to use in the output file. Options are `mm`, `m`, `in` (default), or `tenths` (tenths of an inch). |
| `--models-dir <output model directory>` | Name of output directory to copy component models into. If not used, component models are embedded into the output file. |
| `--models-relative` | With `--models-dir`, use relative paths in the output file. |

## PCB export: XAO

`pcb export xao` 命令将电路板设计导出为 XAO（SALOME/Gmsh）3D 模型文件。

Usage: `kicad-cli pcb export xao [--help] [--output OUTPUT_FILE] [--define-var KEY=VALUE]… [--force] [--no-unspecified] [--no-dnp] [--variant VAR]… [--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] [--no-extra-pad-thickness] [--min-distance MIN_DIST] [--net-filter VAR] [--user-origin VAR] INPUT_FILE`

位置参数：

| | |
|---|---|
| `INPUT_FILE` | Board file to export. |

可选参数：

| | |
|---|---|
| `-h`, `--help` | Show help for the XAO 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 `.xao` 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. |
| `--variant <variant name>` | The name(s) of the variant(s) to output. Can be used multiple times to output multiple variants. When specifying multiple variants, use `${VARIANT}` in the output path to generate separate files for each variant. When `--variant` is not used, the default variant is output. |
| `--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. |

| | |
|---|---|
| --include-soldermask | Include solder mask layers in export as a set of flat faces. |
| --fuse-shapes | Fuse overlapping geometry together in export (time consuming). |
| --fill-all-vias | Don’t cut via holes in conductor layers. |
| --no-extra-pad-thickness | Disable adding additional metal thickness to pads. When not used, pads have 0.005mm added to their metal thickness, which causes pads to be separate faces in the exported model, distinct from the surrounding metal. |
| --min-distance <min distance> | Tolerance for considering two points to be in the same location. Default: 0.01mm. |
| --net-filter <net filter> | Only include copper items belonging to nets matching this wildcard. |
| --user-origin <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 import

The `pcb import` command imports a non-KiCad PCB file to KiCad format. Layers in the input board file are automatically mapped to KiCad layers.

Usage: `kicad-cli pcb import [--help] [--output OUTPUT_FILE] [--format FORMAT] [--layer-map FILE] [--auto-map] [--report-format FORMAT] [--report-file FILE] INPUT_FILE`

位置参数：

| | |
|---|---|
| INPUT_FILE | Non-KiCad format board file to import. |

可选参数：

| | |
|---|---|
| `-h`, `--help` | Show help for the import 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 `.kicad_pcb` file extension. |
| `--format <format>` | The input board file format. Options are `auto` (default), `pads`, `altium`, `eagle`, `cadstar`, `fabmaster`, `pcad`, and `solidworks`. If the format is `auto`, or if no format is given, KiCad will attempt to automatically determine the input board file format. |
| `--report-format <format>` | Report file format. Options are `none` (default), `json`, or `text`. |
| `--report-file <report filename>` | Output filename for the generated import report. When `--report-file` is not used, the report is printed to stdout. |

## PCB 渲染

`pcb render` 命令会生成电路板 3D 模型的光线追踪渲染图，并将其保存为 PNG 或 JPEG 文件。

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] [--use-board-stackup-colors VAR] [--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：

| | |
|---|---|
| `-h`, `--help` | Show help for the render command. |
| `-o <output filename>`, `--output <output filename>` | 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 `.png` (for PNG files) or `.jpg` / `.jpeg` (for JPG files). |
| `-D <variable name>=<value>`, `--define-var <variable_name>=<value>` | Add or override project variable definitions. Can be used multiple times to define multiple variables. |
| `-w <width>`, `--width <width>` | Image width in pixels. Default: 1600. |
| `-h <height>`, `--height <height>` | Image height in pixels. Default: 900. |
| `--side <side>` | The side of the board to render. Options are `top` (default), `bottom`, `left`, `right`, `front`, or `back`. |

| | |
|---|---|
| `--background <background>` | Image background. Options are `default` (default), `transparent`, or `opaque`. For PNG files, `default` is `transparent`. For JPG files, `default` is `opaque`. |
| `--quality <quality>` | Render quality. Options are `basic` (default), `high`, `user`. When `user` is specified, the render settings stored in the project are used. |
| `--preset <color preset>` | Color preset. Options are `follow_pcb_editor`, `follow_plot_settings` (default), or `legacy_preset_flag`. |
| `--use-board-stackup-colors` | Colors defined in the board stackup override colors from the preset. |
| `--floor` | Enables floor, shadows and post-processing, even if disabled in quality preset. |
| `--perspective` | Use perspective projection instead of orthogonal. |
| `--zoom <zoom level>` | Camera zoom factor as an integer. Default: 1. |
| `--pan <camera pan>` | Set camera pan location, in millimeters, with the format `'X,Y,Z'`, e.g. `'3,0,0'`. |
| `--pivot <pivot>` | Set pivot point relative to the board center in centimeters, with the format `'X,Y,Z'` e.g. `'-10,2,0'`. |
| `--rotate <rotation>` | Set board rotation around pivot point, in degrees, with the format `'X,Y,Z'`, e.g. `'-45,0,45'` for isometric view. |
| `--light-top <intensity>` | Top light intensity, format `'R,G,B'` or a single number, range: 0-1. |
| `--light-bottom <intensity>` | Bottom light intensity, format `'R,G,B'` or a single number, range: 0-1. |
| `--light-side <intensity>` | Side lights intensity, format `'R,G,B'` or a single number, range: 0-1. |
| `--light-camera <intensity>` | Camera light intensity, format `'R,G,B'` or a single number, range: 0-1. |
| `--light-side-elevation <elevation>` | Side lights elevation angle in degrees, range: 0-90. |

## PCB upgrade

The `pcb upgrade` command converts a KiCad board file from a previous KiCad board file format to the native format for the current version of KiCad. If the input board file is already in the current file format, no action is taken.

Usage: `kicad-cli pcb upgrade [--help] [--force] INPUT_FILE`

位置参数：

| | |
|---|---|
| `INPUT_FILE` | Board file to upgrade. |

可选参数：

| | |
|---|---|
| `-h`, `--help` | Show help for the upgrade command. |
| `--force` | Re-save the input board file even if it is already in the current file format. |

# 原理图命令

sch 命令可运行电气规则检查、将原理图导出为各种其他文件格式，或导出 BOM 或网表。每个子命令都有自己的选项。

## 原理图 ERC

sch erc 命令对原理图进行电气规则检查并生成报告。

Usage: 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 | Schematic file to run ERC on. |

可选参数：

| | |
|---|---|
| -h, --help | Show help for the ERC command. |
| -o <output filename>, --output <output filename> | Output filename for the generated ERC report. When --output is not used, the output filename will be the same as the input file, with the .rpt or .json file extension, depending on the selected format. |
| -D <variable name>=<value>, --define-var <variable_name>=<value> | Add or override project variable definitions. Can be used multiple times to define multiple variables. |
| --format <format> | Report file format. Options are report (default) or json. |
| --units <unit> | Units to use in the report. Options are mm (default), in, or mils. |
| --severity-all | Report all ERC violations. This is equivalent to using all of the other ERC severity options. |
| --severity-error | Report all error-level ERC violations. This can be combined with the other ERC severity options. |
| --severity-warning | Report all warning-level ERC violations. This can be combined with the other ERC severity options. |
| --severity-exclusions | Report all excluded ERC violations. This can be combined with the other ERC severity options. |
| --exit-code-violations | 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. |

## Schematic export: bill of materials

`sch export bom` 命令从原理图导出 BOM。BOM 导出有多个选项可用于控制格式和包含的字段。此导出方法相当于从符号字段表导出 BOM (导出 BOM)。

| | |
|---|---|
| **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] [--variant VAR]… [--preset PRESET] [--format-preset FMT_PRESET] [--fields FIELDS] [--labels LABELS] [--group-by GROUP_BY] [--sort-field SORT_BY] [--sort-asc VAR] [--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` | Schematic file to export. |

可选参数：

| | |
|---|---|
| `-h`, `--help` | Shows help message and exits |
| `-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 a `.csv` file extension. |
| `--variant <variant name>` | The name(s) of the variant(s) to output. Can be used multiple times to output multiple variants. When specifying multiple variants, use `${VARIANT}` in the output path to generate separate files for each variant. When `--variant` is not used, the default variant is output. |
| `--preset <preset>` | Use a named BOM preset setting from the schematic, e.g. `"Grouped By Value"`. |
| `--format-preset <format preset>` | Use a named BOM format preset setting from the schematic, e.g. `CSV`. |
| `--fields <fields>` | An ordered list of fields to export. `*` includes all fields. Virtual BOM symbol fields such as DNP or Exclude from board can be accessed with `${DNP}` or `${EXCLUDE_FROM_BOARD}`, respectively (see the BOM export documentation for a list of fields). These fields can be specified in this argument with or without the `${}` syntax. Default: "Reference,Value,Footprint,QUANTITY,DNP". |
| `--labels <labels>` | An ordered list of labels to apply the exported fields (default: `"Refs,Value,Footprint,Qty,DNP"`). |
| `--group-by <fields>` | Fields to group references by when field values match. |
| `--sort-field <fields>` | Field name to sort by (default: `"Reference"`). |
| `--sort-asc` | If given, sort in ascending order. If not given, sort in descending order. |

| | |
|---|---|
| --filter <filter> | If given, only components with reference designators that match the given filter string are included in the output. The filter supports wildcards: * matches any number of any characters, including none, and ? matches any single character. |
| --exclude-dnp | Exclude symbols with the "Do not populate" attribute. |
| --include-excluded-from-bom | Include symbols marked "Exclude from BOM". This argument is deprecated as of KiCad 10.0 and has no effect. |
| --field-delimiter <delimiter> | Separator between output fields/columns (default: ","). |
| --string-delimiter <delimiter> | Character to surround fields with (none by default). |
| --ref-delimiter <delimiter> | Character to place between individual references (default: ","). |
| --ref-range-delimiter <delimiter> | Character to place in ranges of references (default: "-"). Leave blank for no ranges. |
| --keep-tabs | Keep tab characters from input fields. Stripped by default. |
| --keep-line-breaks | Keep line break characters from input fields. Stripped by default. |

## Schematic export: DXF

命令 sch export dxf 将原理图导出到 DXF 文件。设计中的每张图纸都会导出到各自的文件中。

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

位置参数：

| | |
|---|---|
| INPUT_FILE | Schematic file to export. |

可选参数：

| | |
|---|---|
| `-h`, `--help` | Show help for the DXF 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. |
| `--variant <variant name>` | The name(s) of the variant(s) to output. Can be used multiple times to output multiple variants. When specifying multiple variants, use `${VARIANT}` in the output path to generate separate files for each variant. When `--variant` is not used, the default variant is output. |
| `-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 DXF without a drawing sheet. |
| `--default-font <font name>` | Default font name. Default: `"KiCad Font"`. |
| `--draw-hop-over` | Draw hop-overs at wire crossings. |
| `--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. |

## Schematic export: HPGL

**NOTE** `kicad-cli sch export hpgl` is not functional in KiCad 10.0.

The `sch export hpgl` command is not functional in KiCad 10.0 as KiCad no longer supports HPGL output. In previous versions of KiCad it exported a schematic to an HPGL file. It is included as a non-functional command for compatibility reasons. It will be removed in a future version of KiCad.

Usage: `kicad-cli sch export hpgl [--help] [--output OUTPUT_DIR] [--drawing-sheet SHEET_PATH] [--define-var KEY=VALUE]… [--variant VAR]… [--theme THEME_NAME] [--black-and-white] [--exclude-drawing-sheet] [--default-font VAR] [--draw-hop-over] [--pages PAGE_LIST] [--pen-size PEN_SIZE] [--origin ORIGIN] INPUT_FILE`

## Schematic export: netlist

命令 `sch export netlist` 将原理图中的网表以 各种格式 导出。

Usage: `kicad-cli sch export netlist [--help] [--output OUTPUT_FILE] [--variant VAR]… [--format FORMAT] INPUT_FILE`

位置参数：

| | |
|---|---|
| `INPUT_FILE` | Schematic file to export. |

可选参数：

| | |
|---|---|
| `-h`, `--help` | Show help for the netlist 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 a `.net` file extension. |
| --variant <variant name> | The name(s) of the variant(s) to output. Can be used multiple times to output multiple variants. When specifying multiple variants, use `${VARIANT}` in the output path to generate separate files for each variant. When `--variant` is not used, the default variant is output. |
| --format <format> | The netlist output format. Options are `kicadsexpr` (default), `kicadxml`, `cadstar`, `orcadpcb2`, `spice`, `spicemodel`, `pads`, or `allegro`. |

## Schematic export: PDF

`sch export pdf` 命令将原理图导出到 PDF 文件。设计中的每个图纸都会导出到 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]… [--variant VAR]… [--theme THEME_NAME] [--black-and-white] [--exclude-drawing-sheet] [--default-font VAR] [--draw-hop-over] [--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` | Schematic file to export. |

可选参数：

| | |
|---|---|
| `-h`, `--help` | Show help for the PDF 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 a `.pdf` file extension. |
| `--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. |
| `--variant <variant name>` | The name(s) of the variant(s) to output. Can be used multiple times to output multiple variants. When specifying multiple variants, use `${VARIANT}` in the output path to generate separate files for each variant. When `--variant` is not used, the default variant is output. |
| `-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 PDF without a drawing sheet. |
| `--default-font <font name>` | Default font name. Default: `"KiCad Font"`. |
| `--draw-hop-over` | Draw hop-overs at wire crossings. |
| `--exclude-pdf-property-popups` | Do not generate property popups in PDF. |
| `--exclude-pdf-hierarchical-links` | Do not generate clickable links for hierarchical elements in PDF. |
| `--exclude-pdf-metadata` | Do not generate PDF metadata from AUTHOR and SUBJECT variables. |
| `-n`, `--no-background-color` | Export schematic without a background color, regardless of theme. |
| `--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. |

## Schematic export: PostScript

命令 `sch export ps` 将原理图导出到 PostScript 文件。设计中的每个图纸都会导出到各自的文件中。

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

位置参数：

| | |
|---|---|
| INPUT_FILE | Schematic file to export. |

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. |
| --variant <variant name> | The name(s) of the variant(s) to output. Can be used multiple times to output multiple variants. When specifying multiple variants, use ${VARIANT} in the output path to generate separate files for each variant. When --variant is not used, the default variant is output. |
| -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". |
| --draw-hop-over | Draw hop-overs at wire crossings. |
| -n, --no-background-color | Export schematic without a background color, regardless of theme. |
| --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. |

## Schematic export: bill of materials (legacy BOM scripts)

命令 sch export python-bom 可从原理图导出 XML BOM 文件。然后，可使用自定义脚本或 原理图 BOM 导出文档 中描述的脚本之一，将 XML BOM 文件处理成所需的 BOM 格式。

Usage: `kicad-cli sch export python-bom [--help] [--output OUTPUT_FILE] INPUT_FILE`

位置参数：

| | |
|---|---|
| INPUT_FILE | Schematic file to export. |

可选参数：

| | |
|---|---|
| -h , --help | Show help for the BOM 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 a -bom.xml suffix and file extension. |

## Schematic export: SVG

命令 sch export svg 将原理图导出到 SVG 文件。设计中的每个图纸都会导出到各自的文件中。

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

位置参数：

| | |
|---|---|
| INPUT_FILE | Schematic file to export. |

可选参数：

| | |
|---|---|
| `-h`, `--help` | Show help for the SVG file export command. |
| `-o <output dir>`, `--output <output dir>` | The output folder for the exported files. 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. |
| `--variant <variant name>` | The name(s) of the variant(s) to output. Can be used multiple times to output multiple variants. When specifying multiple variants, use `${VARIANT}` in the output path to generate separate files for each variant. When `--variant` is not used, the default variant is output. |
| `-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 SVG without a drawing sheet. |
| `--default-font <font name>` | Default font name. Default: `"KiCad Font"`. |
| `--draw-hop-over` | Draw hop-overs at wire crossings. |
| `-n`, `--no-background-color` | Export schematic without a background color, regardless of theme. |
| `--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. |

## Schematic upgrade

The `sch upgrade` command converts a KiCad schematic file from a previous KiCad schematic file format to the native format for the current version of KiCad. If the input schematic file is already in the current file format, no action is taken.

| | |
|---|---|
| **NOTE** | Only the specified schematic file is upgraded. If the schematic file contains any child sheets, the child sheets are not upgraded. |

Usage: `kicad-cli sch upgrade [--help] [--force] INPUT_FILE`

位置参数：

| | |
|---|---|
| `INPUT_FILE` | Schematic file to upgrade. |

可选参数：

| | |
|---|---|
| `-h`, `--help` | Show help for the upgrade command. |
| `--force` | Re-save the input schematic file even if it is already in the current file format. |

# 符号命令

`sym` 子命令用于将符号导出到另一种格式，或将符号库升级到 KiCad 符号文件格式的当前版本。

## 符合导出

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

Usage: `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` | Symbol library file to use for export. |

可选参数：

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

支持的输入符号格式包括：

- KiCad 符号库（`.kicad_sym`）
- KiCad（6.0之前版本）符号库（`.lib`）
- Altium 原理图库（`.SchLib`）
- Altium 集成库(`.IntLib`)
- CADSTAR 元件库（`.lib`）

EAGLE XML 库 ( `.lbr` )

- EasyEDA (JLCEDA) 标准版文件 ( `.json` )
- EasyEDA (JLCEDA) 专业版文件 ( `.elibz` , `.epro` , `.zip` )

Usage: `kicad-cli sym upgrade [--help] [--output OUTPUT_FILE_OR_DIR] [--force] INPUT_FILE_OR_DIR`

位置参数：

| | |
|---|---|
| `INPUT_FILE_OR_DIR` | Symbol or symbol library to upgrade. This can be an unpacked symbol ( `.kicad_sym` file containing a single symbol), an unpacked symbol library (folder containing `.kicad_sym` files), or a packed symbol library ( `.kicad_sym` file containing multiple symbols). |

可选参数：

| | |
|---|---|
| `-h` , `--help` | Show help for the upgrade command. |
| `-o <output file or directory>` , `--output <output file or directory>` | The output file or directory for the upgraded symbol library. When the output path is a file, the symbols are saved as a single-file ("packed") `.kicad_sym` library. When the output path is a folder, the symbols are saved as individual ("unpacked") `.kicad_sym` files in the folder, with one file per symbol. When `--output` is not used, the upgraded symbol library is saved over the original library. |
| `--force` | Re-save the input library even if it is already in the current file format. |

# 版本命令

`version` 命令用于显示 KiCad 的版本信息。若不带任何参数，它将直接显示版本号，例如 `7.0.7`。您还可以通过使用 `--format` 参数以其他格式显示版本信息。

**NOTE** 使用 `kicad-cli version --format about` 命令获取版本信息，并在提交 GitLab 上的 bug 报告或功能请求时包含此信息。

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

可选参数：

| | |
|---|---|
| `-h`, `--help` | Show help for the version command. |
| `--format <format>` | Format of the version number. Options are `plain` (default), `commit`, or `about`. `plain` prints the version number (e.g. `7.0.7`), which is the default if the `--format` argument is not used. `commit` prints the hash of the git commit for the build of KiCad you are using. `about` prints the full version information, including library versions and basic system information. You can use the `about` version information in bug reports. |