KiCad Command-Line Interface

The KiCad Team

Table of Contents

Reference manual

Contributors

Graham Keeth

Feedback

The KiCad project welcomes feedback, bug reports, and suggestions related to the software or its documentation. For more information on how to submit feedback or report an issue, please see the instructions at https://www.kicad.org/help/report-an-issue/

Introduction to the KiCad Command-Line Interface

KiCad provides a command-line interface, which is available by running the `kicad-cli` binary. With the command-line interface, you can perform a number of actions on schematics, PCBs, symbols, and footprints in an automated fashion, such as plotting Gerber files from a PCB design or upgrading a symbol library from a legacy file format to a modern format.

NOTE	On macOS, the `kicad-cli` executable is located at `/Applications/KiCad/KiCad.app/Contents/MacOS/kicad-cli`.

The `kicad-cli` command has 5 subcommands: `fp`, `pcb`, `sch`, `sym`, and `version`. Each subcommand may have its own subcommands and arguments. For example, to export Gerber files from a PCB you could run `kicad-cli pcb export gerbers example.kicad_pcb`.

You can add the `--help` or `-h` flag to see information about each subcommand. For example, running `kicad-cli pcb -h` prints usage information about the `pcb` subcommand, and `kicad-cli pcb export gerbers -h` prints usage information specifically for the `pcb export gerbers` subcommand.

Footprint commands

The `fp` subcommand exports footprints to another format or upgrades the footprint libraries to the current version of the KiCad footprint file format.

Footprint export

The `fp export svg` command exports one or more footprints from the specified library into SVG files.

Usage: `kicad-cli fp export svg [--help] [--output OUTPUT_FILE] [--layers LAYER_LIST] [--define-var KEY=VALUE] [--theme VAR] [--footprint FOOTPRINT_NAME] [--black-and-white] INPUT_DIR`

Positional arguments:

INPUT_DIR	Footprint library directory to export (`.pretty`).

Optional arguments:

`-h`, `--help`	Show help for the footprint SVG export command.
`-o <output dir>`, `--output <output dir>`	The output directory for the exported files. When this argument 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.
`-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.
`--black-and-white`	Export footprints in black and white.

Footprint upgrade

The `fp upgrade` command upgrades the the specified footprint library from a legacy 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.

Usage: `kicad-cli fp upgrade [-h] [--output OUTPUT_DIR] [--force] INPUT_DIR`

Positional arguments:

INPUT_DIR	Footprint library directory to upgrade (`.pretty`).

Optional arguments:

`-h`, `--help`	Show help for the footprint upgrade command.
`-o <output dir>`, `--output <output dir>`	The output directory for the upgraded footprints. When this argument 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.

PCB commands

The `pcb` command runs a design rule check or exports a board to various other file formats, including fabrication and 3D files.

PCB DRC

The `pcb drc` command runs a design rule check on a board and generates a report.

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] INPUT_FILE`

Positional arguments:

INPUT_FILE	Board file to run DRC on.

`-h`, `--help`	Show help for the DRC command.
-o <output filename>, `--output <output filename>	Output filename for the generated DRC report. When this argument 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.

PCB drill file export

The `pcb export drill` command exports a drill file from a board.

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] [--map-format MAP_FORMAT] [--gerber-precision VAR] INPUT_FILE`

Positional arguments:

`INPUT_FILE`	Board file to export.

Optional arguments:

`-h`, `--help`	Show help for the drill file export command.
`-o <output dir>`, `--output <output dir>`	The output directory for the drill file. When this argument is not used, the drill file is 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.
`--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 DXF export

The `pcb export dxf` command exports a board design to a DXF file.

Usage: `kicad-cli pcb export dxf [--help] [--output OUTPUT_FILE] [--layers LAYER_LIST] [--drawing-sheet SHEET_PATH] [--define-var KEY=VALUE] [--exclude-refdes] [--exclude-value] [--use-contours] [--include-border-title] [--output-units UNITS] INPUT_FILE`

Positional arguments:

`INPUT_FILE`	Board file to export.

Optional arguments:

`-h`, `--help`	Show help for the DXF export command.
`-o <output filename>`, `--output <output filename>`	The output filename. When this argument is not used, the output filename will be the same as the input file, with the `.dxf` 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.
`-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.
`--erd`, `--exclude-refdes`	Exclude footprint reference designators from plot.
`--ev`, `--exclude-value`	Exclude footprint values from plot.
`--uc`, `--use-contours`	Plot graphic items using their contours.
`-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).

PCB Gerber export: one layer per file

The `pcb export gerbers` command exports a board design to Gerber files, with one layer per file.

> **NOTE**
>
> Be aware that there are two distinct Gerber export commands, `gerber` and `gerbers`. The `gerber` command plots multiple PCB layers to a single Gerber file, while the `gerbers` command plots multiple Gerber files, with one PCB layer per file. The `gerbers` command is typically the correct command to use for having a PCB fabricated.

Usage: `kicad-cli pcb export gerbers [--help] [--output OUTPUT_FILE] [--layers LAYER_LIST] [--drawing-sheet SHEET_PATH] [--define-var KEY=VALUE] [--exclude-refdes] [--exclude-value] [--`

include-border-title] [--no-x2] [--no-netlist] [--subtract-soldermask] [--disable-aperture-macros] [--use-drill-file-origin] [--precision PRECISION] [--no-protel-ext] [--common-layers COMMON_LAYER_LIST] [--board-plot-params] INPUT_FILE

Positional arguments:

INPUT_FILE	Board file to export.

Optional arguments:

-h, --help	Show help for the Gerber export command.
-o <output dir>, --output <output dir>	The output folder for the exported files. When this argument 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.
--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.
--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.).
--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.
--board-plot-params	Use the Gerber plot settings already configured in the board file.

PCB Gerber export: multiple layers per file

The `pcb export gerber` command exports one or more board layers to a single Gerber file.

NOTE	Be aware that there are two distinct Gerber export commands, `gerber` and `gerbers`. The `gerber` command plots multiple PCB layers to a single Gerber file, while the `gerbers` command plots multiple Gerber files, with one PCB layer per file. The `gerbers` command is typically the correct command to use for having a PCB fabricated.

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

Positional arguments:

`INPUT_FILE`	Board file to export.

Optional arguments:

`-h`, `--help`	Show help for the Gerber export command.
`-o <output filename>`, `--output <output filename>`	The output filename. When this argument is not used, the output filename will be the same as the input file, with the `.gbr` file extension.
`-l <layer list>`, `--layers <layer list>`	A comma-separated list of layer names to plot from the board, such as `F.Cu,B.Cu`. All layers will be plotted in the output file. At least one layer must be given.
`--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.
`--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.).

PCB GLB export

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

Usage: `kicad-cli pcb export glb [--help] [--output OUTPUT_FILE] [--define-var KEY=VALUE] [--force] [--grid-origin] [--drill-origin] [--no-unspecified] [--no-dnp] [--subst-models] [--board-only] [--include-tracks] [--include-zones] [--min-distance MIN_DIST] [--user-origin VAR] INPUT_FILE`

Positional arguments:

`INPUT_FILE`	Board file to export.

Optional arguments:

-h , --help	Show help for the GLB export command.
-o <output filename> , --output <output filename>	The output filename. When this argument 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.
--grid-origin	Use grid origin as origin of output file.
--drill-origin	Use drill origin as origin of 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.
--subst-models	Replace VRML models in footprints with STEP models of the same name, if they exist.
--board-only	Only include the board itself in the generated model; exclude all component models.
--include-tracks	Include tracks in export (time consuming).
--include-zones	Include zones in export (time consuming).
--min-distance <min distance>	Tolerance for considering two points to be in the same location. Default: 0.01mm.
--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 IPC-2581 export

The pcb export ipc2581 command exports a board design in IPC-2581 format.

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] INPUT_FILE

INPUT_FILE	Board file to export.

Optional arguments:

`-h`, `--help`	Show help for the IPC-2581 export command.
`-o <output filename>`, `--output <output filename>`	The output filename. When this argument 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 3.
`--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.

PCB PDF export

The `pcb export pdf` command exports a board design to a PDF file.

Usage: `kicad-cli pcb export pdf [--help] [--output OUTPUT_FILE] [--layers LAYER_LIST] [--drawing-sheet SHEET_PATH] [--define-var KEY=VALUE] [--mirror] [--exclude-refdes] [--exclude-value] [--include-border-title] [--negative] [--black-and-white] [--theme THEME_NAME] [--drill-shape-opt VAR] INPUT_FILE`

Positional arguments:

`INPUT_FILE`	Board file to export.

Optional arguments:

`-h`, `--help`	Show help for the PDF export command.
`-o <output filename>`, `--output <output filename>`	The output filename. When this argument is not used, the output filename will be the same as the input file, with the `.pdf` file extension.
`-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.
`--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.
`-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).

PCB position file export

The `pcb export pos` command exports a position file from a board design.

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] INPUT_FILE`

Positional arguments:

`INPUT_FILE`	Board file to export.

Optional arguments:

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

PCB STEP export

The `pcb export step` command exports a board design to a STEP file.

Usage: `kicad-cli pcb export step [--help] [--output OUTPUT_FILE] [--define-var KEY=VALUE] [--force] [--grid-origin] [--drill-origin] [--no-unspecified] [--no-dnp] [--subst-models] [--board-only] [--include-tracks] [--include-zones] [--min-distance MIN_DIST] [--no-optimize-step] [--user-origin VAR] INPUT_FILE`

Positional arguments:

`INPUT_FILE`	Board file to export.

Optional arguments:

`-h`, `--help`	Show help for the STEP file export command.
`-o <output filename>`, `--output <output filename>`	The output filename. When this argument 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.
`--grid-origin`	Use grid origin as origin of output file.
`--drill-origin`	Use drill origin as origin of 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.
`--subst-models`	Replace VRML models in footprints with STEP models of the same name, if they exist.
`--board-only`	Only include the board itself in the generated model; exclude all component models.
`--include-tracks`	Include tracks in export (time consuming).
`--include-zones`	Include zones in export (time consuming).
`--min-distance <min distance>`	Tolerance for considering two points to be in the same location. Default: `0.01mm`.
`--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 SVG export

The `pcb export svg` command exports a board design to an SVG file.

Usage: `kicad-cli pcb export svg [--help] [--output OUTPUT_FILE] [--layers LAYER_LIST] [--drawing-sheet SHEET_PATH] [--define-var KEY=VALUE] [--mirror] [--theme THEME_NAME] [--negative] [--black-and-white] [--page-size-mode MODE] [--exclude-drawing-sheet] [--drill-shape-opt SHAPE_OPTION] INPUT_FILE`

Positional arguments:

`INPUT_FILE`	Board file to export.

Optional arguments:

-h, --help	Show help for the SVG file export command.
-o <output filename>, --output <output filename>	The output filename. When this argument is not used, the output filename will be the same as the input file, with the .svg file extension.
-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.
--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.
-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.
--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.
--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).

PCB VRML export

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

Usage: `kicad-cli pcb export vrml [--help] [--output OUTPUT_FILE] [--define-var KEY=VALUE] [--force] [--user-origin VAR] [--units VAR] [--models-dir VAR] [--models-relative] INPUT_FILE`

Positional arguments:

INPUT_FILE	Board file to export.

Optional arguments:

-h, --help	Show help for the VRML export command.
-o <output filename>, --output <output filename>	The output filename. When this argument 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.
--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.

Schematic commands

The `sch` command runs an electrical rule check, exports a schematic to various other file formats, or exports a bill of materials or netlist. Each subcommand has its own options.

Schematic ERC

The `sch erc` command runs an electrical rule check on a schematic and generates a report.

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`

Positional arguments:

INPUT_FILE	Schematic file to run ERC on.

-h, --help	Show help for the ERC command.
-o <output filename>, `--output <output filename>	Output filename for the generated ERC report. When this argument 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 bill of materials export

The `sch export bom` command exports a BOM from a schematic. The BOM export has a number of options for controlling the format and included fields. This export method is equivalent to exporting a BOM from the symbol fields table.

NOTE	To export a BOM using the legacy XML and Python BOM script workflow, use the `sch export python-bom` command.

Usage: `kicad-cli sch export bom [--help] [--output OUTPUT_FILE] [--preset PRESET] [--format-preset FMT_PRESET] [--fields FIELDS] [--labels LABELS] [--group-by GROUP_BY] [--sort-field SORT_BY] [--sort-asc] [--filter FILTER] [--exclude-dnp] [--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`

Positional arguments:

`INPUT_FILE`	Schematic file to export.

Optional arguments:

`-h`, `--help`	Shows help message and exits
`-o <output filename>`, `--output <output filename>`	The output filename. When this argument is not used, the output filename will be the same as the input file, with a `.csv` file extension.
`--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. Special symbol fields such as DNP or Exclude from board can be accessed with `${DNP}` or `${EXCLUDE_FROM_BOARD}`, respectively (see the text variable documentation for a list of fields). 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>`	Filter string to remove output lines.
`--exclude-dnp`	Exclude symbols with the "Do not populate" attribute.
`--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 DXF export

The `sch export dxf` command exports a schematic to a DXF file. Each sheet in the design is exported to its own file.

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

Positional arguments:

INPUT_FILE	Schematic file to export.

Optional arguments:

-h, --help	Show help for the DXF file export command.
-o <output dir>, --output <output dir>	The output folder for the exported files. When this argument is not used, the files are exported to the current directory.
--drawing-sheet <sheet path>	Path to drawing sheet to use in plot, overriding the drawing sheet specified in the schematic file.
-D <variable name>=<value>, --define-var <variable_name>=<value>	Add or override project variable definitions. Can be used multiple times to define multiple variables.
-t <theme name>, --theme <theme name>	The name of the theme to use for export. If no theme is given, the schematic editor's currently selected theme is used.
-b, --black-and-white	Export schematic in black and white.
-e, --exclude-drawing-sheet	Plot DXF without a drawing sheet.
--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 HPGL export

The `sch export hpgl` command exports a schematic to an HPGL file for a pen plotter. Each sheet in the design is exported to its own file.

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

Positional arguments:

INPUT_FILE	Schematic file to export.

Optional arguments:

`-h`, `--help`	Show help for the HPGL file export command.
`-o <output dir>`, `--output <output dir>`	The output folder for the exported files. When this argument 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.
`-e`, `--exclude-drawing-sheet`	Plot HPGL without a drawing sheet.
`--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.
`-p <pen size>`, `--pen-size <pen size>`	Set the pen width. The default pen size is 0.5 mm.
`-r <origin>`, `--origin <origin>`	Set plotter origin and scale. Options are `0`, `1` (default), `2`, or `3`. `0` sets the origin to the bottom left and uses plotter units. `1` sets the origin to the center and uses plotter units. `2` scales to the page, and `3` scales to the content within the page.

Schematic netlist export

The `sch export netlist` command exports a netlist in various formats from a schematic.

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

Positional arguments:

`INPUT_FILE`	Schematic file to export.

Optional arguments:

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

Schematic PDF export

The `sch export pdf` command exports a schematic to a PDF file. Each sheet in the design is exported to its own page in the PDF file.

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

Positional arguments:

INPUT_FILE	Schematic file to export.

Optional arguments:

-h, --help	Show help for the PDF file export command.
-o <output filename>, --output <output filename>	The output filename. When this argument 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.
-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.
--exclude-pdf-property-popups	Do not generate property popups in PDF.
-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 PostScript export

The sch export ps command exports a schematic to a PostScript file. Each sheet in the design is exported to its own file.

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

Positional arguments:

INPUT_DIR	Schematic file to export.

Optional arguments:

`-h`, `--help`	Show help for the PS file export command.
`-o <output dir>`, `--output <output dir>`	The output folder for the exported files. When this argument is not used, the files are exported to the current directory.
`--drawing-sheet <sheet path>`	Path to drawing sheet to use in plot, overriding the drawing sheet specified in the schematic file.
`-D <variable name>=<value>`, `--define-var <variable_name>=<value>`	Add or override project variable definitions. Can be used multiple times to define multiple variables.
`-t <theme name>`, `--theme <theme name>`	The name of the theme to use for export. If no theme is given, the schematic editor's currently selected theme is used.
`-b`, `--black-and-white`	Export schematic in black and white.
`-e`, `--exclude-drawing-sheet`	Plot PS without a drawing sheet.
`-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 bill of materials export (legacy BOM scripts)

The `sch export python-bom` command exports an XML BOM file from a schematic. The XML BOM file can then be processed into your desired BOM format using a custom script or one of the scripts described in the schematic BOM export documentation.

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

Positional arguments:

`INPUT_FILE`	Schematic file to export.

Optional arguments:

`-h`, `--help`	Show help for the BOM export command.
`-o <output filename>`, `--output <output filename>`	The output filename. When this argument is not used, the output filename will be the same as the input file, with a `-bom.xml` suffix and file extension.

Schematic SVG export

The `sch export svg` command export a schematic to an SVG file. Each sheet in the design is exported to its own file.

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

Positional arguments:

`INPUT_FILE`	Schematic file to export.

Optional arguments:

`-h`, `--help`	Show help for the SVG file export command.
`-o <output dir>`, `--output <output dir>`	The output folder for the exported files. When this argument is not used, the files are exported to the current directory.
`--drawing-sheet <sheet path>`	Path to drawing sheet to use in plot, overriding the drawing sheet specified in the schematic file.
`-D <variable name>=<value>`, `--define-var <variable_name>=<value>`	Add or override project variable definitions. Can be used multiple times to define multiple variables.
`-t <theme name>`, `--theme <theme name>`	The name of the theme to use for export. If no theme is given, the schematic editor’s currently selected theme is used.
`-b`, `--black-and-white`	Export schematic in black and white.
`-e`, `--exclude-drawing-sheet`	Plot SVG without a drawing sheet.
`-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.

Symbol commands

The `sym` subcommand exports symbols to another format or upgrades symbol libraries to the current version of the KiCad symbol file format.

Symbol export

The `sym export svg` command exports one or more symbols from the specified library into SVG files.

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`

Positional arguments:

`INPUT_FILE`	Symbol library file to use for export.

Optional arguments:

`-h`, `--help`	Show help for the symbol SVG export command.
`-o <output dir>`, `--output <output dir>`	The output folder for the exported files. When this argument 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.

Symbol upgrade

The `sym upgrade` command upgrades the the specified symbol library from a legacy 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.

Usage: `kicad-cli sym upgrade [--help] [--output OUTPUT_FILE] [--force] INPUT_FILE`

Positional arguments:

`INPUT_FILE`	Symbol library to upgrade.

Optional arguments:

`-h`, `--help`	Show help for the symbol upgrade command.
`-o <output filename>`, `--output <output filename>`	The output filename for the upgraded symbol library. When this argument 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.

Version commands

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

> **NOTE** Use `kicad-cli version --format about` for version information to include when submitting bug reports or feature requests on Gitlab.

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

Optional arguments:

`--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.