KiCad Befehlszeilenschnittstelle

The KiCad Team

Table of Contents

KiCad 10.0 Referenzhandbuch

Copyright

Mitwirkende

Graham Keeth

Feedback

Das KiCad-Projekt freut sich über Rückmeldungen, Fehlerberichte und Vorschläge in Bezug auf die Software oder ihre Dokumentation. Weitere Informationen zum Einreichen von Feedback oder zum Melden eines Problems finden Sie in den Anweisungen unter https://www.kicad.org/help/report-an-issue/

Version der Software und Dokumentation

Dieses Benutzerhandbuch basiert auf KiCad 10.0.0. Funktionalität und Aussehen können sich in anderen Versionen von KiCad unterscheiden.

Revision der Dokumentation: `90e4b228`.

Einführung in die KiCad Befehlszeilenschnittstelle

KiCad bietet eine Befehlszeilenschnittstelle, die durch Ausführen der Binärdatei `kicad-cli` verfügbar ist. Mit der Befehlszeilenschnittstelle können Sie eine Reihe von Aktionen für Schaltpläne, Leiterplatten, Symbole und Footprints automatisiert ausführen, z. B. das Plotten von Gerber-Dateien aus einem Leiterplattenentwurf oder das Aktualisieren einer Symbolbibliothek von einem älteren Dateiformat auf ein modernes Format.

NOTE	Unter macOS befindet sich die Anwendung `kicad-cli` unter `/Applications/KiCad/KiCad.app/Contents/MacOS/kicad-cli`.

`kicad-cli` verfügt über 6 Unterbefehle: `fp`, `jobset`, `pcb`, `sch`, `sym` und `version`. Jeder Unterbefehl kann eigene Unterbefehle und Argumente haben. Um beispielsweise Gerber-Dateien aus einer Leiterplatte zu exportieren, könnten Sie `kicad-cli pcb export gerbers example.kicad_pcb` ausführen.

Sie können das Flag `--help` oder `-h` hinzufügen, um Informationen zu den einzelnen Unterbefehlen anzuzeigen. Wenn Sie beispielsweise `kicad-cli pcb -h` ausführen, werden Informationen zur Verwendung des Unterbefehls `pcb` angezeigt, und wenn Sie `kicad-cli pcb export gerbers -h` ausführen, werden Informationen zur Verwendung speziell für den Unterbefehl `pcb export gerbers` angezeigt.

Footprint-Befehle

Der Unterbefehl `fp` exportiert Footprints in ein anderes Format oder aktualisiert die Footprint-Bibliotheken auf die aktuelle Version des KiCad-Footprint-Dateiformats.

Footprint-Export

Der Befehl `fp export svg` exportiert einen oder mehrere Footprints aus der angegebenen Bibliothek in SVG-Dateien.

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`

Positionsargumente:

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

Optionale Argumente:

`-h`, `--help`	Hilfe zum Befehl zum Exportieren von Footprint-SVG-Dateien anzeigen.
`-o <Ausgabeverzeichnis>`, `--output <Ausgabeverzeichnis>`	Das Ausgabeverzeichnis für die exportierten Dateien. Für jede Ebene jedes Footprints in der Bibliothek wird eine Datei ausgegeben. Wenn `--output` nicht verwendet wird, werden die Dateien in das aktuelle Verzeichnis exportiert.
`-l <Ebenenliste>`, `--layers <Ebenenliste>`	Eine durch Kommas getrennte Liste der Ebenennamen, die aus dem Footprint exportiert werden sollen, z. B. `F.Cu,B.Cu`. Wenn keine Ebenen angegeben sind, werden alle Ebenen exportiert. Layernamen können als kanonische Layernamen (`F.Cu`, `In.1`, `F.Fab` usw.) oder als benutzerdefinierte Layernamen angegeben werden, wobei benutzerdefinierte Layernamen zuerst abgeglichen werden.
`-D <Variablenname>=<Wert>`, `--define-var <Variablenname>=<Wert>`	Projektvariablendefinitionen hinzufügen oder überschreiben. Kann mehrfach verwendet werden, um mehrere Variablen zu definieren.
`-t <Themenname>`, `--theme <Themenname>`	Der Name des Themas, das für den Export verwendet werden soll. Wenn kein Thema angegeben ist, wird das derzeit im Footprinteditor ausgewählte Thema verwendet.
`--fp <Footprint>`, `--footprint <Footprint>`	Der Name des spezifischen Footprints, der aus der Bibliothek exportiert werden soll. Wenn dieses Argument nicht verwendet wird, werden alle Footprints in der Bibliothek exportiert.
`--sp`, `--sketch-pads-on-fab-layers`	Pad-Umrisse und deren Nummern auf den vorderen und hinteren Fab-Layern zeichnen.
`--hdnp`, `--hide-DNP-footprints-on-fab-layers`	Text und Grafiken von DNP-Footprints nicht auf Fab-Layern plotten.
`--sdnp`, `--sketch-DNP-footprints-on-fab-layers`	Grafiken von DNP-Footprints im Skizzenmodus auf Fab-Layern zeichnen.
`--cdnp`, `--crossout-DNP-footprints-on-fab-layers`	Ein „X" über den Hof von DNP-Footprints auf Fab-Layern zeichnen und deren Referenzbezeichnungen durchstreichen.
`--black-and-white`	Footprints in Schwarz-Weiß exportieren.

Footprint-Upgrade

Der Befehl `fp upgrade` konvertiert die angegebene Footprintbibliothek aus einem älteren KiCad-Footprint-Format oder einem Nicht-KiCad-Footprint-Format in das native Format der aktuellen KiCad-Version. Befindet sich die Eingabebibliothek bereits im aktuellen Dateiformat, wird keine Aktion durchgeführt.

Unterstützte Footprint-Eingabeformate sind:

- KiCad-Footprintbibliothek (Ordner `.pretty` mit `.kicad_mod`-Dateien)
- KiCad (vor 5.0) Footprintbibliothek (`.mod`, `.emp`)
- Altium-Footprintbibliothek (`.PcbLib`)

Altium integrierte Bibliothek (`.IntLib`)

- CADSTAR PCB-Archiv (`.cpa`)
- EAGLE XML-Bibliothek (`.lbr`)
- EasyEDA (JLCEDA) Std-Datei (`.json`)
- EasyEDA (JLCEDA) Pro-Datei (`.elibz` , `.epro` , `.zip`)
- GEDA/PCB-Bibliothek (Ordner mit `.fp` -Dateien)

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

Positionsargumente:

`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).

Optionale Argumente:

`-h` , `--help`	Hilfe zum Upgrade-Befehl anzeigen.
`-o <Ausgabeverzeichnis>` , `--output <Ausgabeverzeichnis>`	Das Ausgabeverzeichnis für die aktualisierten Footprints. Wenn `--output` nicht verwendet wird, werden die ursprünglichen Footprints mit den aktualisierten Footprints überschrieben.
`--force`	Die Eingabebibliothek erneut speichern, auch wenn sie bereits im aktuellen Dateiformat vorliegt.

Jobset-Befehle

Der Befehl `jobset run` führt einen vordefinierten Jobset aus.

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

Positionsargumente:

`INPUT_FILE`	Projektdatei, die mit dem Jobset verwendet werden soll.

Optionale Argumente:

`-h`, `--help`	Hilfe für den Befehl „jobset“ anzeigen.
`--stop-on-error`	Da die Jobs nacheinander ausgeführt werden, wird die Ausführung nach dem Fehlschlagen eines Jobs beendet. Wenn diese Option nicht angegeben wird, werden die Jobs nach dem Fehlschlagen eines Jobs weiter ausgeführt.
`-f <Jobset-Datei>`, `--file <Jobset-Datei>`	Die auszuführende Jobset-Datei (`.kicad_jobset`).
`--output <Zielbeschreibung oder ID>`	Das zu generierende Jobset-Ziel. Wenn kein Ziel angegeben ist, werden alle Ziele generiert. Das Ziel wird durch seine Beschreibung oder seine eindeutige ID angegeben. Die angegebene Beschreibung muss eindeutig sein. Wenn der Jobset mehr als ein Ziel mit der angegebenen Beschreibung enthält, wird keines davon ausgeführt. IDs sind von Natur aus eindeutig und können verwendet werden, um auf ein Ziel zu verweisen, auch wenn die Beschreibung des Ziels nicht eindeutig ist. Die ID für jedes Ziel wird vom Befehl `jobset run` ausgegeben, wenn `--output` nicht verwendet wird. Sie ist auch in der Datei `.kicad_jobset` unter dem Schlüssel `id` des Ziels zu finden.

Platinenkommandos

Der Befehl „pcb“ führt eine Designregelprüfung durch oder exportiert eine Platine in verschiedene andere Dateiformate, darunter Fertigungs- und 3D-Dateien.

PCB DRC

Der Befehl „pcb drc“ führt eine Designregelprüfung auf einer Platine durch und erstellt einen Bericht.

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`

Positionsargumente:

`INPUT_FILE`	Platinendatei, auf der DRC ausgeführt werden soll.

Optionale Argumente:

`-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

Der Befehl „pcb export 3dpdf“ exportiert ein Leiterplattendesign in eine PDF-Datei, die ein eingebettetes 3D-Modell der Leiterplatte enthält.

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`

Positionsargumente:

INPUT_FILE	Zu exportierende Platinen-Datei.

Optionale Argumente:

-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)

Der Befehl „pcb export brep“ exportiert ein Leiterplattendesign in eine BREP-3D-Modelldatei (OCCT-native Boundary Representation).

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`

Positionsargumente:

`INPUT_FILE`	Zu exportierende Platinen-Datei.

Optionale Argumente:

`-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

Der Befehl „pcb export drill“ exportiert eine Bohrdatei aus einer Platine.

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`

Positionsargumente:

`INPUT_FILE`	Zu exportierende Platinen-Datei.

Optionale Argumente:

-h, --help	Hilfe für den Befehl zum Exportieren von Bohrdateien anzeigen.
-o <Ausgabeverzeichnis>, --output <Ausgabeverzeichnis>	Das Ausgabeverzeichnis für die Bohrdatei(en). Wenn --output nicht verwendet wird, werden die Bohrdatei(en) im aktuellen Verzeichnis gespeichert.
--format <Format>	Das Bohrdateiformat. Optionen sind excellon (Standard) oder gerber.
--drill-origin <Ursprung>	Der Koordinatenursprung für die Bohrdatei. Optionen sind absolute (Standard) zur Verwendung des absoluten Ursprungs der Platine oder plot zur Verwendung des Bohr-/Platzierungsursprungs der Platine.
--excellon-zeros-format <Format>	Das Nullformat für die Bohrdatei. Optionen sind decimal (Standard), suppressleading, suppresstrailing oder keep. Gilt nur für Bohrdateien im Excellon-Format.
--excellon-oval-format <Format>	Den Bohrmodus für ovale Löcher steuern. Optionen sind route und alternate (Standard). Gilt nur für Bohrdateien im Excellon-Format.
-u <Einheit>, --excellon-units <Einheit>	Die Einheit für die Bohrdatei. Optionen sind mm (Standard) oder in. Gilt nur für Bohrdateien im Excellon-Format.
--excellon-mirror-y	Die Bohrdatei in Y-Richtung spiegeln. Gilt nur für Bohrdateien im Excellon-Format.
--excellon-min-header	Einen minimalen Header in der Bohrdatei verwenden. Gilt nur für Bohrdateien im Excellon-Format.
--excellon-separate-th	Separate Bohrdateien für durchkontaktierte und nicht durchkontaktierte Bohrungen erstellen. Gilt nur für Bohrdateien im Excellon-Format.
--generate-map	Zusätzlich zur Bohrdatei eine Kartendatei erstellen.
--generate-report	Eine Berichtdatei mit allen Bohrpunkten erstellen.
--report-path <report filename>	Der Ausgabedateiname für die Bohrberichtdatei. Wenn --report-path nicht verwendet wird, entspricht der Dateiname des Berichts dem der Eingabedatei, mit dem Suffix und der Dateiendung -drill.rpt.
--generate-tenting	Separate Bohrdateien für überzeltete Bohrungen generieren. Gilt nur für Bohrdateien im Gerber X2-Format.
--map-format <Format>	Das Format der Map-Datei. Optionen sind pdf (Standard), gerberx2, ps, dxf oder svg.
--gerber-precision <precision>	Die Genauigkeit (Anzahl der Stellen) für die Bohrdatei. Gültige Optionen sind 5 oder 6 (Standard). Gilt nur für Bohrdateien im Gerber-Format.

PCB export: DXF

Der Befehl „pcb export dxf“ exportiert ein Leiterplattendesign in eine DXF-Datei.

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`

Positionsargumente:

`INPUT_FILE`	Zu exportierende Platinen-Datei.

Optionale Argumente:

`-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

Der Befehl `pcb export gencad` exportiert ein Leiterplattendesign in eine GenCAD-Datei.

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`

Positionsargumente:

INPUT_FILE	Zu exportierende Platinen-Datei.

Optionale Argumente:

`-h`, `--help`	Hilfe für den DXF-Exportbefehl anzeigen.
`-o <Ausgabedateiname>`, `--output <Ausgabedateiname>`	Der Ausgabedateiname. Wenn `--output` nicht verwendet wird, entspricht der Ausgabedateiname dem Namen der Eingabedatei mit der Dateiendung `.cad`.
`-D <Variablenname>=<Wert>`, `--define-var <Variablenname>=<Wert>`	Projektvariablendefinitionen hinzufügen oder überschreiben. Kann mehrfach verwendet werden, um mehrere Variablen zu definieren.
`-f`, `--flip-bottom-pads`	Untere Footprint-Padstacks umdrehen.
`--unique-pins`	Eindeutige Pin-Namen generieren.
`--unique-footprints`	Erzeugt eine neue Form für jede Footprint-Instanz (Formate werden nicht wiederverwendet).
`--use-drill-origin`	Ursprung der Bohr-/Platzierungsdatei als Ursprung verwenden.
`--store-origin-coord`	Ursprungskoordinaten in der Datei speichern.

PCB export: Gerber

Der Befehl `pcb export gerbers` exportiert ein Leiterplattendesign in Gerber-Dateien, wobei jede Datei einen Layer enthält.

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`

Positionsargumente:

INPUT_FILE	Zu exportierende Platinen-Datei.

Optionale Argumente:

`-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

Der Befehl `pcb export glb` exportiert ein Leiterplattendesign in eine GLB-Datei (binäre glTF) für 3D-Modelle.

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`

Positionsargumente:

`INPUT_FILE`	Zu exportierende Platinen-Datei.

Optionale Argumente:

`-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

Der Befehl `pcb export ipc2581` exportiert ein Leiterplattendesign im 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] [--variant VAR]… INPUT_FILE`

Positionsargumente:

`INPUT_FILE`	Zu exportierende Platinen-Datei.

Optionale Argumente:

`-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

Der Befehl `pcb export ipcd356` generiert eine IPC-D-356-Netzliste aus dem Board-Design.

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

Positionsargumente:

INPUT_FILE	Zu exportierende Platinen-Datei.

Optionale Argumente:

`-h`, `--help`	Hilfe für den IPC-D-356-Exportbefehl anzeigen.
`-o <Ausgabedateiname>`, `--output <Ausgabedateiname>`	Der Name der Ausgabedatei. Wenn `--output` nicht verwendet wird, entspricht der Name der Ausgabedatei dem Namen der Eingabedatei mit der Dateierweiterung `.d356`.

PCB export: ODB++

Der Befehl `pcb export odb` exportiert ein Leiterplattendesign im ODB++-Format.

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

Positionsargumente:

INPUT_FILE	Zu exportierende Platinen-Datei.

Optionale Argumente:

`-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

Der Befehl `pcb export pdf` exportiert ein Leiterplattendesign in eine PDF-Datei. Jede Lage kann als eigene Datei oder als Blatt in einer einzelnen Datei ausgegeben werden.

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`

Positionsargumente:

`INPUT_FILE`	Zu exportierende Platinen-Datei.

Optionale Argumente:

`-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

Der Befehl `pcb export ply` exportiert ein Leiterplattendesign in eine PLY-3D-Modelldatei.

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`

Positionsargumente:

`INPUT_FILE`	Zu exportierende Platinen-Datei.

Optionale Argumente:

`-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

Der Befehl `pcb export pos` exportiert eine Positionsdatei aus einem Leiterplattenentwurf.

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`

Positionsargumente:

`INPUT_FILE`	Zu exportierende Platinen-Datei.

Optionale Argumente:

`-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

Der Befehl `pcb export ps` exportiert ein Leiterplattendesign in eine PostScript-Datei.

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`

Positionsargumente:

INPUT_FILE	Zu exportierende Platinen-Datei.

Optionale Argumente:

`-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`

Positionsargumente:

`INPUT_FILE`	Board file to export statistics from.

Optionale Argumente:

`-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

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] [--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`

Positionsargumente:

`INPUT_FILE`	Zu exportierende Platinen-Datei.

Optionale Argumente:

`-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

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

Usage: `kicad-cli pcb export stl [--help] [--output OUTPUT_FILE] [--define-var KEY=VALUE]… [--force] [--no-unspecified] [--no-dnp] [--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`

Positionsargumente:

`INPUT_FILE`	Zu exportierende Platinen-Datei.

Optionale Argumente:

`-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`

Positionsargumente:

`INPUT_FILE`	Zu exportierende Platinen-Datei.

Optionale Argumente:

`-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`

Positionsargumente:

`INPUT_FILE`	Zu exportierende Platinen-Datei.

Optionale Argumente:

`-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

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

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`

Positionsargumente:

`INPUT_FILE`	Zu exportierende Platinen-Datei.

Optionale Argumente:

`-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

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

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

Positionsargumente:

`INPUT_FILE`	Zu exportierende Platinen-Datei.

Optionale Argumente:

`-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

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

Usage: `kicad-cli pcb export xao [--help] [--output OUTPUT_FILE] [--define-var KEY=VALUE]… [--force] [--no-unspecified] [--no-dnp] [--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`

Positionsargumente:

`INPUT_FILE`	Zu exportierende Platinen-Datei.

Optionale Argumente:

`-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`

Positionsargumente:

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

Optionale Argumente:

`-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 render

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

Usage: `kicad-cli pcb render [--help] [--output OUTPUT_FILE] [--define-var KEY=VALUE]… [--width WIDTH] [--height HEIGHT] [--side SIDE] [--background BG] [--quality QUALITY] [--preset PRESET] [--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`

Positionsargumente:

`INPUT_FILE`	Board file to render.

Optionale Argumente:

`-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`

Positionsargumente:

INPUT_FILE	Board file to upgrade.

Optionale Argumente:

`-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.

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`

Positionsargumente:

`INPUT_FILE`	Schematic file to run ERC on.

Optionale Argumente:

`-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

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] [--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`

Positionsargumente:

`INPUT_FILE`	Schematic file to export.

Optionale Argumente:

`-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

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]… [--variant VAR]… [--theme THEME_NAME] [--black-and-white] [--exclude-drawing-sheet] [--default-font VAR] [--draw-hop-over] [--pages PAGE_LIST] INPUT_FILE`

Positionsargumente:

INPUT_FILE	Schematic file to export.

Optionale Argumente:

`-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

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

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

Positionsargumente:

`INPUT_FILE`	Schematic file to export.

Optionale Argumente:

`-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

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]… [--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`

Positionsargumente:

`INPUT_FILE`	Schematic file to export.

Optionale Argumente:

-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

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]… [--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`

Positionsargumente:

INPUT_FILE	Schematic file to export.

Optionale Argumente:

-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)

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`

Positionsargumente:

`INPUT_FILE`	Schematic file to export.

Optionale Argumente:

`-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

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]… [--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`

Positionsargumente:

`INPUT_FILE`	Schematic file to export.

Optionale Argumente:

`-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`

Positionsargumente:

`INPUT_FILE`	Schematic file to upgrade.

Optionale Argumente:

`-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.

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`

Positionsargumente:

`INPUT_FILE`	Symbol library file to use for export.

Optionale Argumente:

`-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.

Symbol upgrade

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

Supported input symbol formats are:

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

- EAGLE XML-Bibliothek (`.lbr`)
- EasyEDA (JLCEDA) Std-Datei (`.json`)
- EasyEDA (JLCEDA) Pro-Datei (`.elibz` , `.epro` , `.zip`)

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

Positionsargumente:

`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).

Optionale Argumente:

`-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.

Version commands

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

NOTE 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]`

Optionale Argumente:

`-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.