CNC_PCB
PCB Milling G-code Generator

User Manual (adapted for GRBL 1.1f)

USER MANUAL

3018 Pro Ultra
CNC Router

2023.5.23 V1.1

Version: generated 2026-01-14

Contents

1. Overview

CNC_PCB converts PCB manufacturing files (Gerber and Excellon) into GRBL-compatible G-code for PCB isolation milling, drilling, and outline cutting. The same project can also generate optional engraving paths (for example, silkscreen).

The program is general-purpose, but its automatic filename handling and layer expectations are adapted to common outputs from KiCad and EasyEDA.

- **Copper isolation (Top)**: offset toolpaths that isolate copper features on the top layer.

- **Drilling**: drills holes from Excellon files and can plan multiple drill sizes.

- **Board outline**: cuts the PCB outline and can mill larger holes/slots using a router tool.

- **Silkscreen engraving (Top)**: engraves silkscreen geometry as shallow lines.

- **Soldermask clear (Pads, Top)**: optional clearing paths over pad openings.

2. Program start and required files

Start the application by running **main.py** or **main.pyw** (depending on the platform packaging).

Load either:

- A **ZIP** exported from a PCB CAD tool, or

- A folder containing Gerber files plus Excellon drill files.

The program can normalize certain EasyEDA exports by copying/renaming files into a consistent set of expected names. If working with other naming schemes, rename files to match the expected layer names described in the Files tab section.

3. Quick start workflow

- Export Gerbers + drills from the PCB CAD tool (ZIP export recommended).

- Open the program and go to **Files** -> **Select Gerber file or ZIP**.

- Verify the detected layers and enabled operations.

- Choose an **Output directory**.

- In **Bits**, assign tools for the enabled operations (create custom tools as needed in **Bits Editor**).

- In **Job Options**, set PCB thickness, safe heights, passes, and drilling planner settings.

- Use **Preview** to confirm orientation, origin, and toolpaths.

- Click **Generate G-code**.

4. User interface tour

The interface is organized into tabs. Typical use is left-to-right: Files -> Bits -> Job Options -> Preview -> Generate.

- **Files**: load the board, inspect detected layers, enable/disable operations, set output behavior.

- **Bits**: select the tool used by each operation and configure drilling selection modes.

- **Bits Editor**: create and manage a tool library saved in *bits.ini*.

- **Job Options**: cutting parameters, drilling planner rules, and safety/parking settings.

- **Preview**: visual check of layers and generated toolpaths.

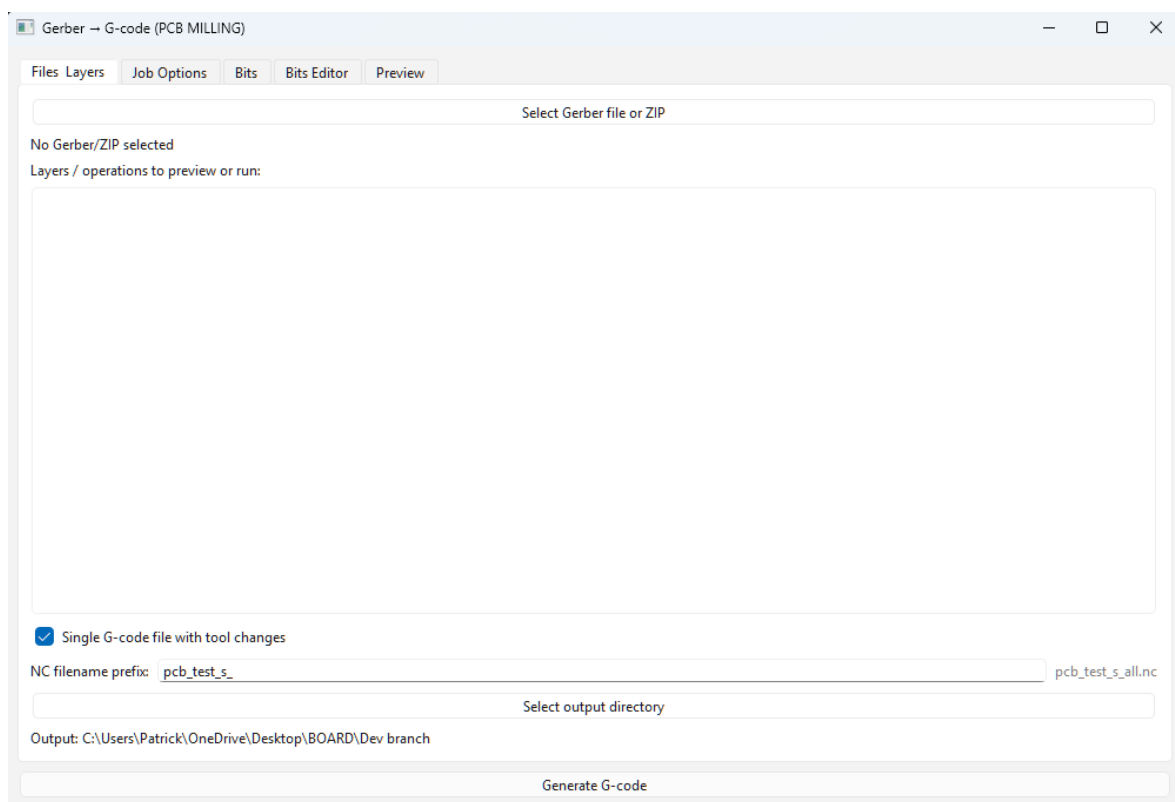- **Generate G-code**: writes the output .nc file(s).
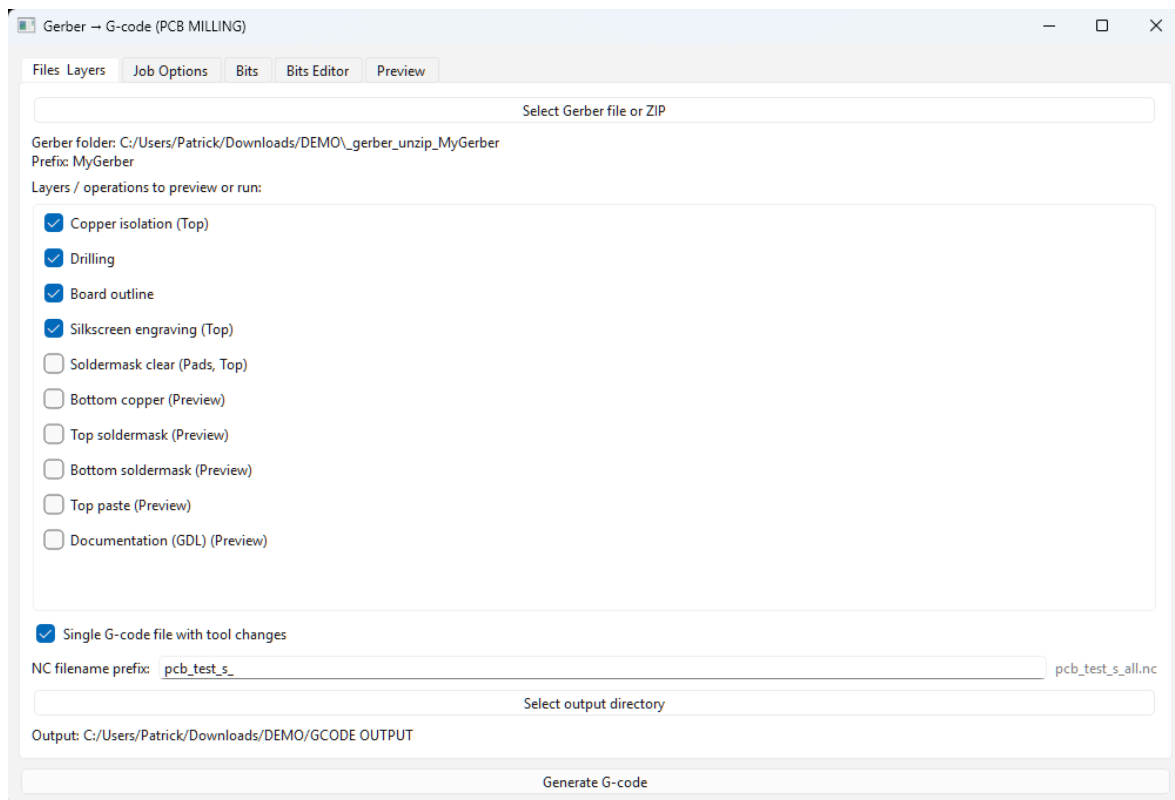

Figure 1 - Main window and tabs.

Figure 2 - A loaded job showing detected layers and selected operations.

4.1 Loaded job information and layer meaning

After loading a ZIP or Gerber folder, the Files tab shows the board prefix and a layer list. The list combines **runnable operations** and **preview-only layers**.

Gerber layers typically represent copper, mask, paste, silkscreen, and the board outline. Excellon layers represent drill hits (PTH and NPTH).

4.1.1 Common layer roles

- **Top copper**: the electrical copper features for the top side. Used as the reference layer for preview bounds and the X/Y origin shift.

- **Bottom copper**: shown for reference in preview; not used for G-code generation in the current operation set.

- **Board outline**: the perimeter cut (and sometimes internal routing). Used for outline cutting and for milling larger holes/slots.

- **Silkscreen**: printed ink artwork; can be engraved as shallow lines.

- **Solder mask**: openings in the mask; used by the soldermask-clear operation if enabled.

- **Paste**: stencil openings; preview-only.

- **Doc / fabrication**: notes, drawings, and reference marks; preview-only.

4.1.2 File naming expectations

The program expects a consistent naming convention once files are in the working folder. EasyEDA exports are commonly normalized into names like:

- *NAME-TopLayer.gbr*, *NAME-BottomLayer.gbr*

- *NAME-TopSilkLayer.gbr*, *NAME-BottomSilkLayer.gbr*

- *NAME-TopSolderMaskLayer.gbr*, *NAME-BottomSolderMaskLayer.gbr*

- *NAME-TopPasteLayer.gbr*, *NAME-BottomPasteLayer.gbr*

- *NAME-BoardOutLine.gbr*, *NAME-DocLayer.gbr*

- *NAME-PTH.drl* and *NAME-NPTH.drl* (drills)

KiCad exports can be used by either exporting with a matching naming scheme or renaming files to the expected names above.

5. Files tab

Use this tab first. It controls: what is loaded, which operations are enabled, and where output is written.
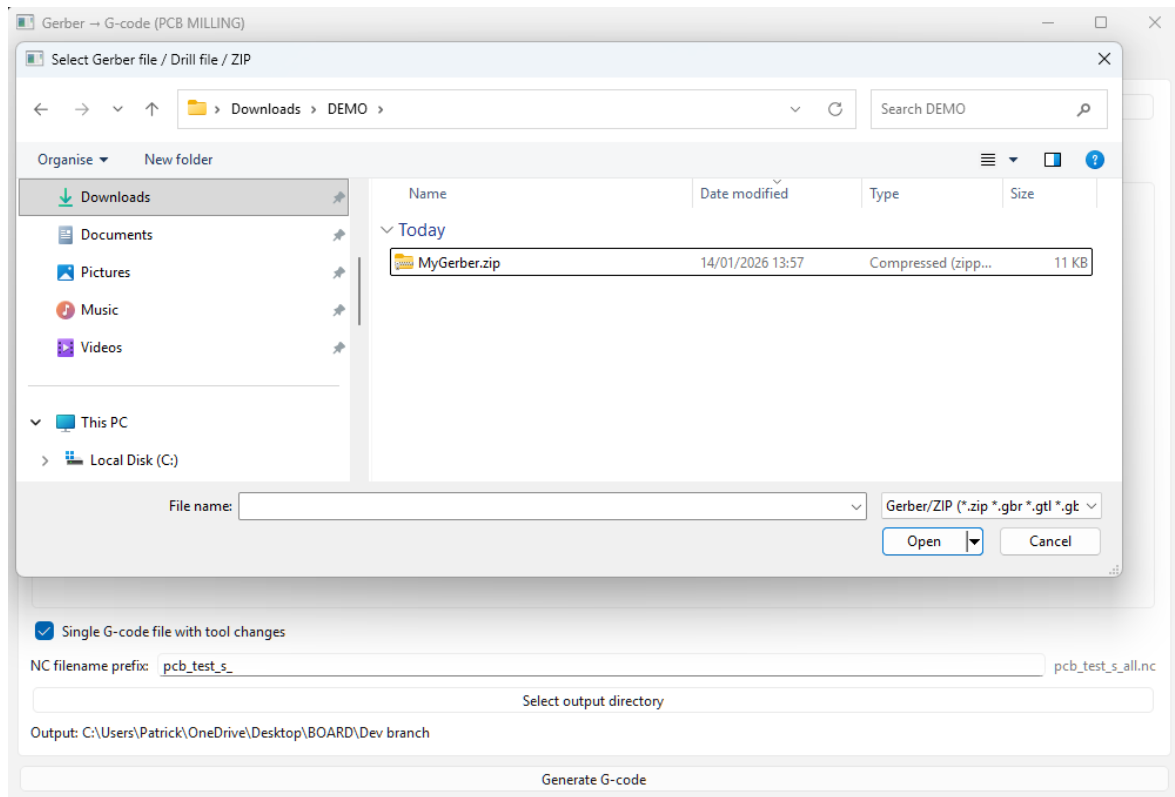

Figure 3 - Selecting a Gerber file, drill file, folder, or ZIP export.

5.1 Selecting a Gerber folder or ZIP

Click **Select Gerber file or ZIP** and choose a ZIP, a Gerber file, or a drill file. If a ZIP contains a single top-level folder, that folder is used automatically.

For certain EasyEDA exports, files may be copied/renamed into the expected naming convention so the rest of the workflow can locate layers reliably.

5.2 Layers and operation list

After loading, a checkbox list appears. Checked items are included when generating G-code. Unchecked items are ignored.

5.2.1 Runnable operations

- **Copper isolation (Top)**
- **Drilling**
- **Board outline**
- **Silkscreen engraving (Top)**
- **Soldermask clear (Pads, Top)**

5.2.2 Preview-only layers

- Bottom copper, bottom silkscreen, bottom soldermask, bottom paste
- Top paste
- Documentation/fabrication layers

Preview-only layers can be used to confirm alignment and orientation even when they are not used for toolpath generation.

5.3 Combined output vs separate files

Single G-code file with tool changes controls whether all enabled operations are appended into one file or written as separate operation files.

- Combined: one file (commonly *all.nc*) with pauses for tool changes between operations.
- Separate: one file per operation (for example *top_copper_isolation.nc*, *drill.nc*, *board_outline.nc*).

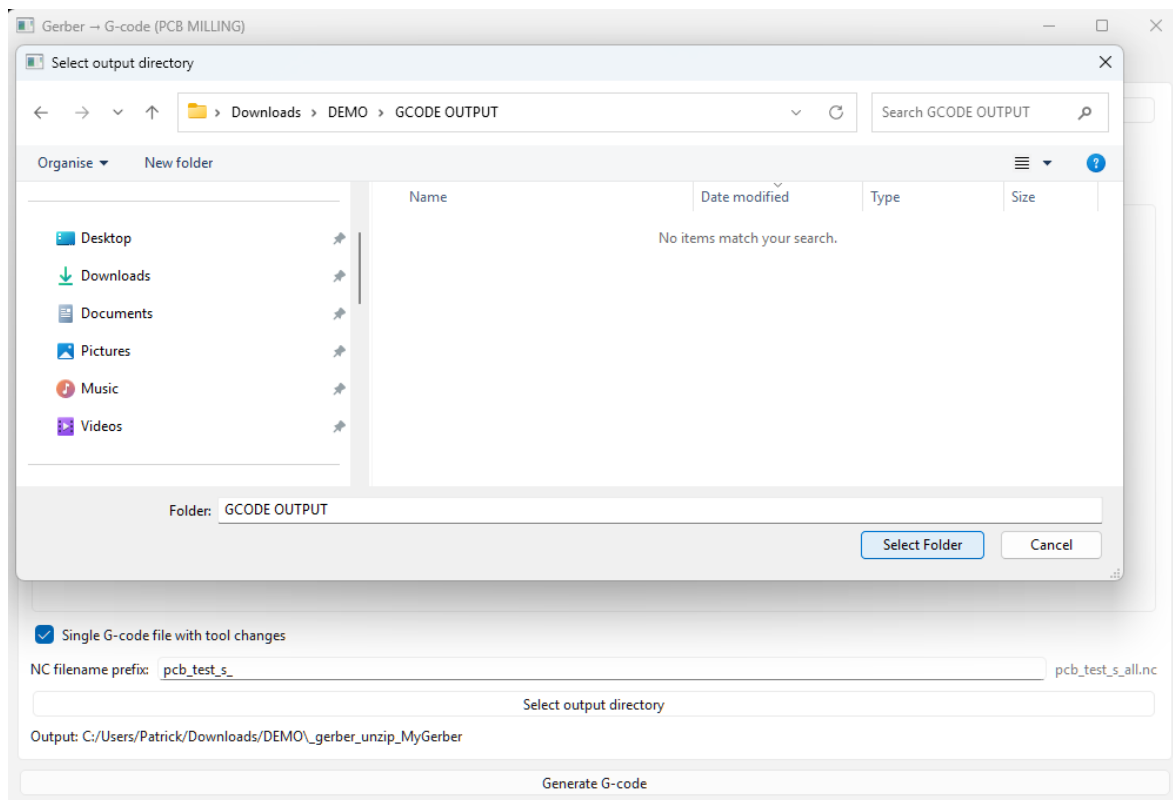## 5.4 Output directory and file prefix

Figure 4 - Output folder selection and filename prefix.

Choose an output directory to keep generated .nc files separate from the Gerbers. If an output directory is not set, output is written to the working Gerber folder.

The **NC filename prefix** is applied to generated files. Characters other than letters, numbers, '_' and '-' are removed.
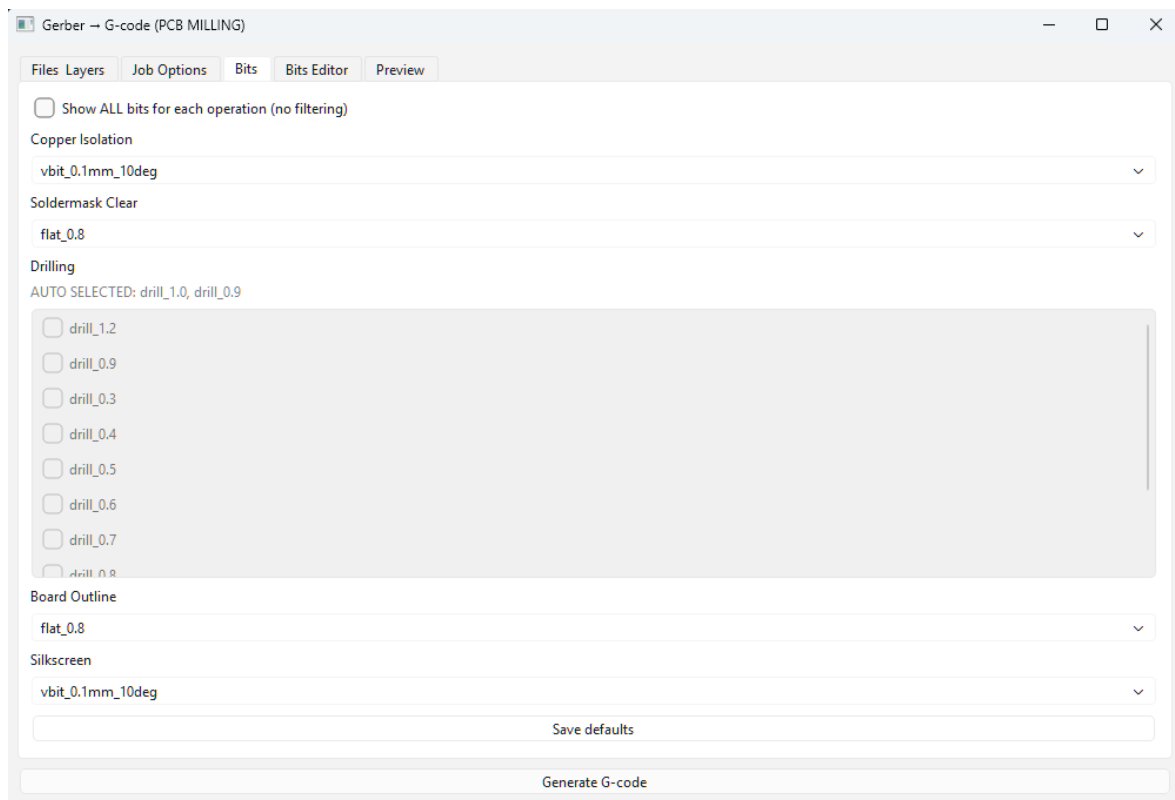
6. Bits tab

Figure 5 - Assigning tools to operations and configuring drilling selection.

This tab maps each enabled operation to a tool definition from **bits.ini**. Add or edit tools in **Bits Editor** to match the cutters available.

6.1 Drill planner modes

Drilling supports multiple drill sizes.

- **AUTO**: selects up to a maximum number of drill sizes automatically.

- **MANUAL**: drill sizes are selected explicitly from the list.

Holes are grouped by selected drill size and ordered to reduce travel moves. Tool-change pauses are inserted between drill sizes when combined output is enabled.

6.2 Show ALL bits

Enable **Show ALL bits** to select any tool for any operation (advanced/override behavior).
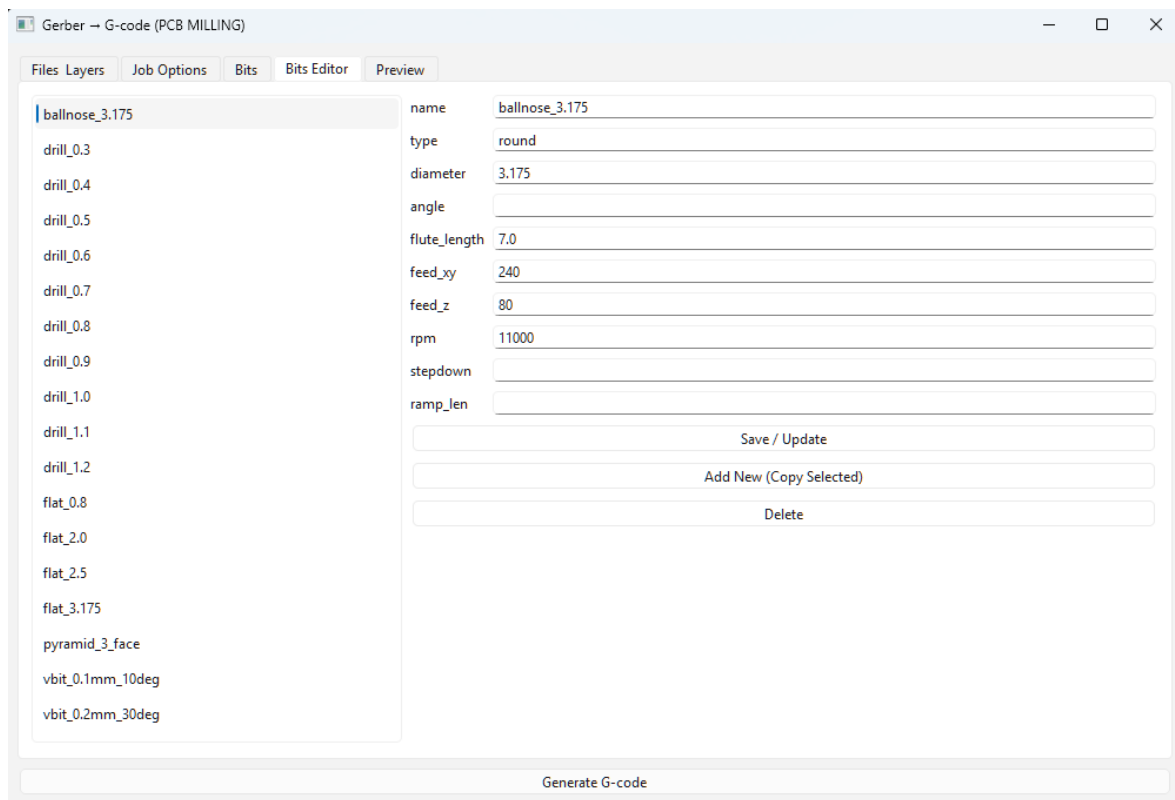
7. Bits Editor tab

Figure 6 - Creating and editing tools in bits.ini.

Tools are stored in **bits.ini**. Each tool section includes fields such as type, diameter, feeds, and RPM. Create tools that match the available cutters and the machine's capabilities.

7.1 Tool fields

- **type**: vbit, drill, flat, round (used for filtering).

- **diameter**: cutter diameter for drills/flat tools; for V-bits it represents the intended isolation cut width reference.

- **angle**: V-bit included angle (degrees).

- **feed_xy**: cutting feed (mm/min).

- **feed_z**: plunge feed (mm/min).

- **rpm**: spindle command value used in G-code (if supported by the controller/spindle).

8. Job Options tab

Job Options controls cutting depths, passes, drill planning rules, and safety/parking moves. The sections below follow the layout shown in the images.
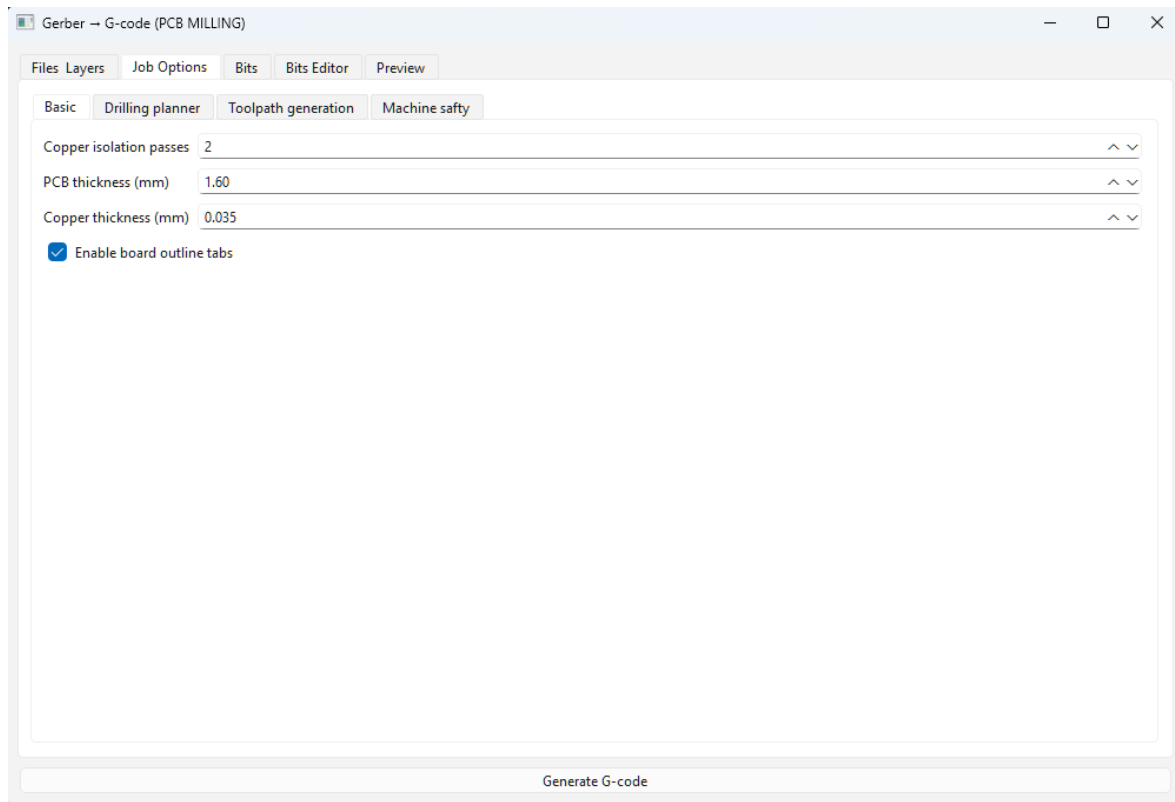

Figure 7 - Job Options: Basic.

8.1 Basic settings

- **Copper isolation passes**: number of offset rings around copper features.

- **PCB thickness**: used for drill depth and through-cut depth (outline/slots/large holes).

- **Copper thickness**: limits isolation depth for V-bits to stay shallow.

- **Enable board outline tabs**: leaves small bridges so the board stays attached during outline cutting.
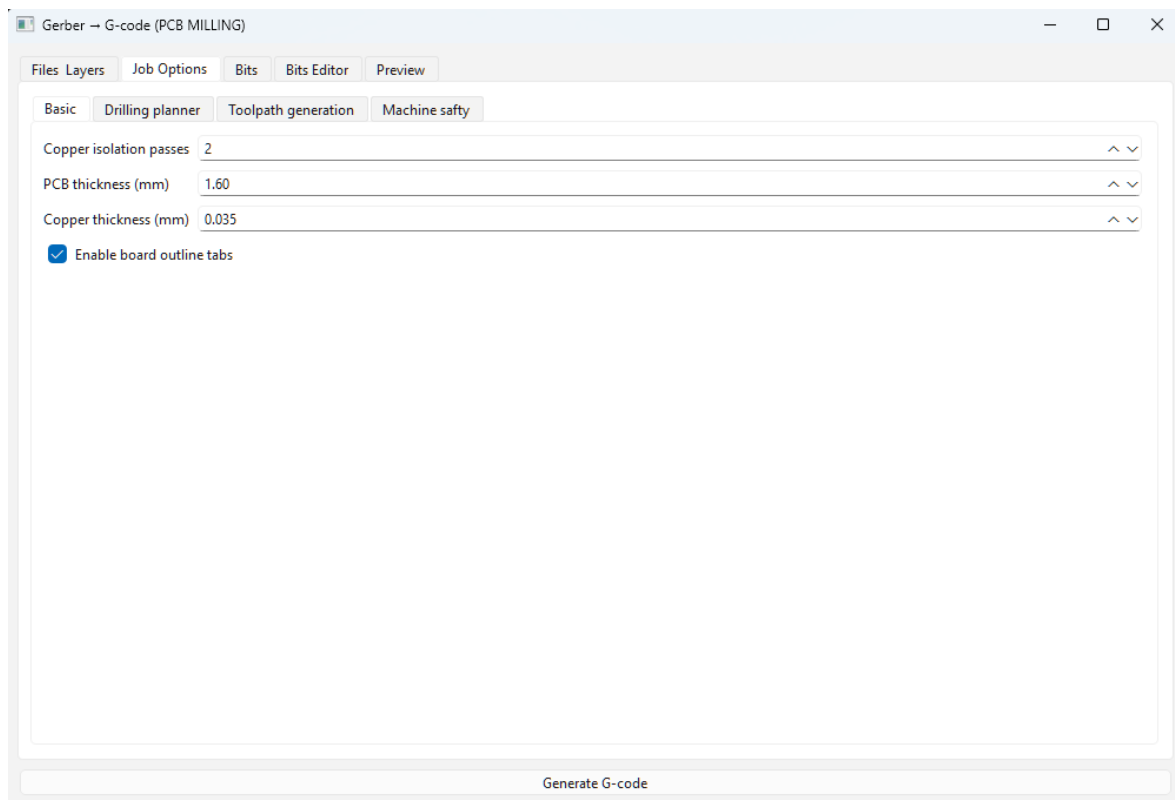
Figure 7 (reference) - Basic options remain the starting point for depths and passes.

8.2 Isolation depth for V-bits

For V-bits, the program computes a geometric depth from the intended cut width and the V angle, then limits the result using the copper thickness setting.

Depth calculation:

depth = (cut_width / 2) / tan(angle / 2)

The computed depth is capped to approximately **copper_thickness + 0.01 mm** to avoid excessive cutting into the substrate.
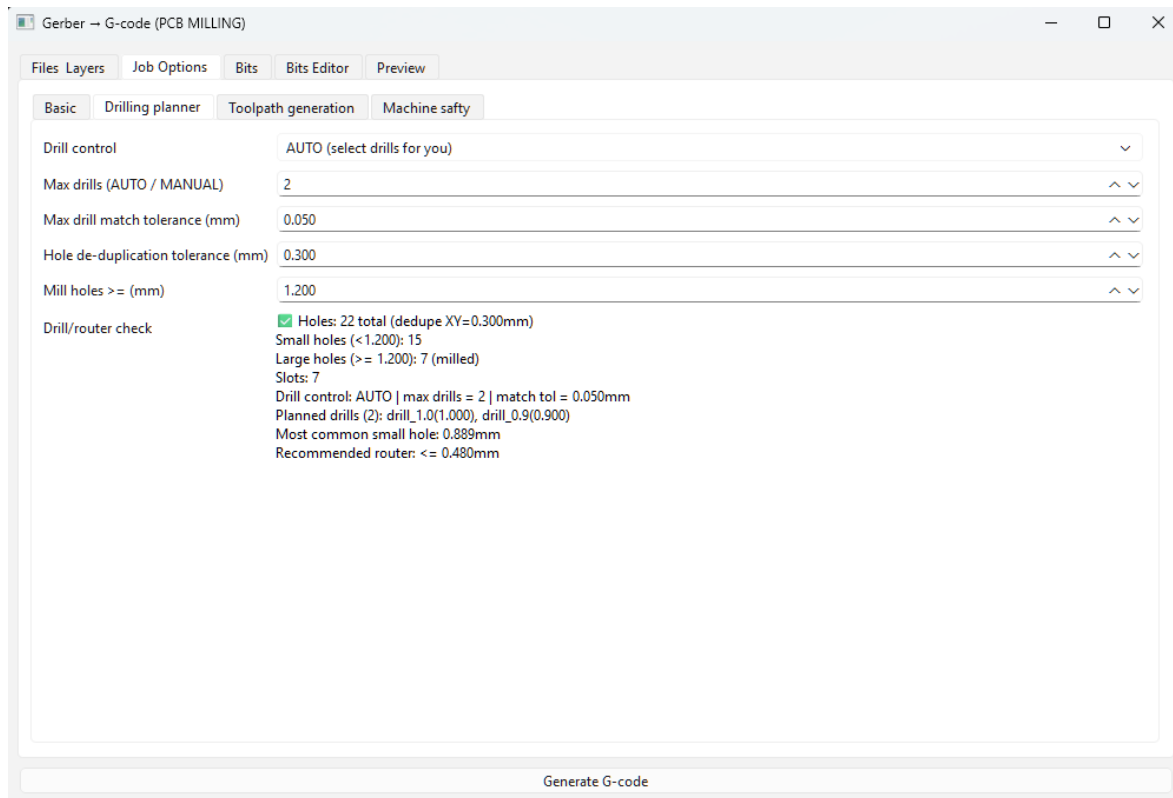
Figure 8 - Job Options: Drilling planner.

8.3 Drilling planner options

- **Drill control**: AUTO or MANUAL.

- **Max drills**: maximum number of drill sizes to use.

- **Max drill match tolerance**: allowed diameter mismatch when assigning drills to holes.

- **Hole de-duplication tolerance**: merges near-duplicate holes within a small radius.

- **Mill holes >=**: holes above this size are milled (using the outline/router tool) instead of drilled.

If warnings indicate a tool is too large for a slot or milled feature, adjust the router tool diameter or the mill-holes threshold.
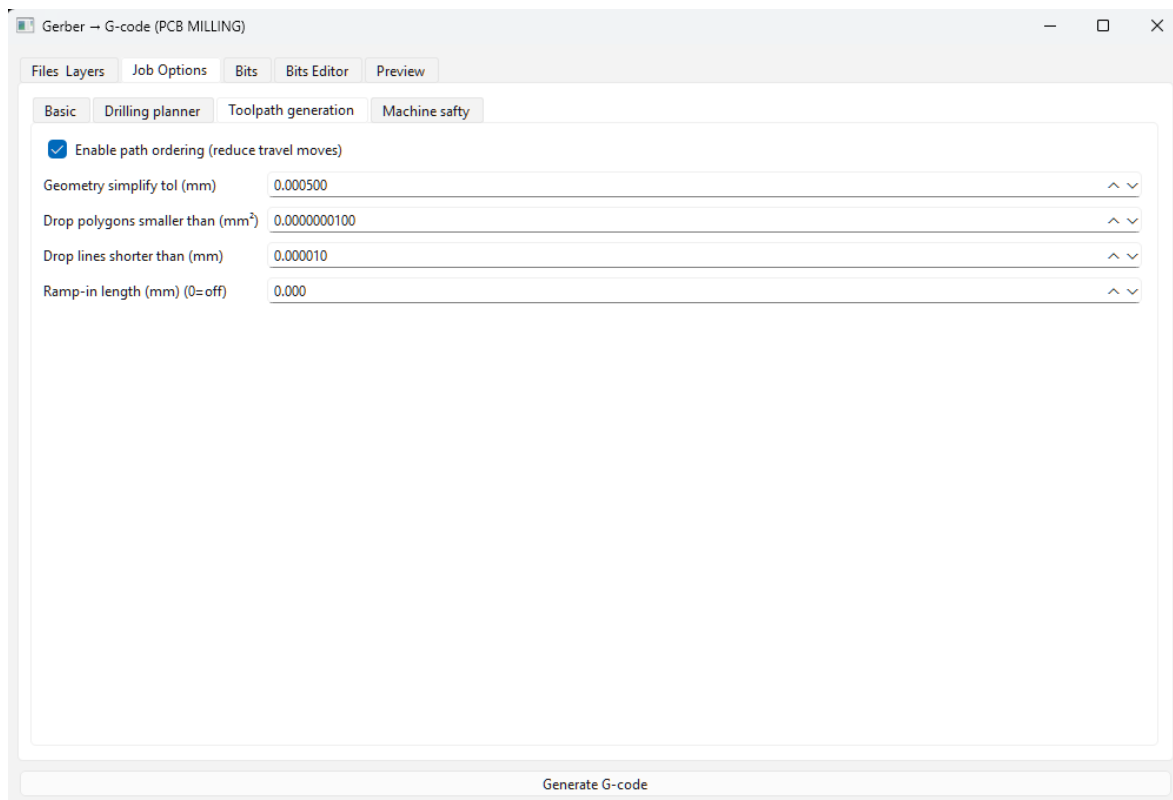
Figure 9 - Job Options: Toolpath generation.

8.4 Toolpath generation options

- **Enable path ordering**: reduces travel moves by reordering segments where safe.

- **Geometry simplify tolerance**: simplifies geometry to reduce tiny segments (use cautiously).

- **Drop polygons smaller than**: filters out tiny polygon artifacts.

- **Drop lines shorter than**: filters out tiny line segments.

- **Ramp-in length**: ramps down on through cuts to reduce tool load.
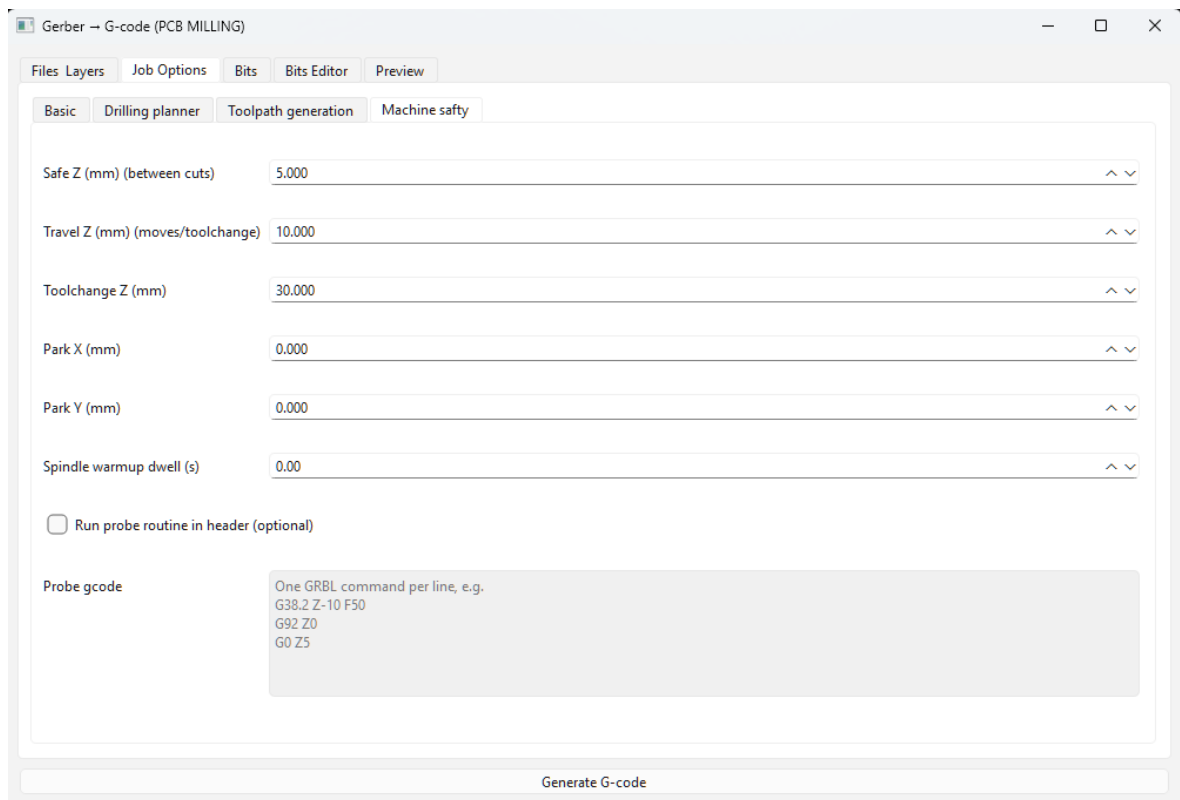
8.5 Machine safety and parking

Figure 10 - Job Options: Machine safety.

- **Safe Z**: height used between nearby cuts.

- **Travel Z**: height for longer moves and after tool changes.

- **Toolchange Z**: height before moving to the tool-change park position.

- **Park X/Y**: parking location for tool changes.

- **Spindle warmup dwell**: pause after starting the spindle before cutting.

- **Probe G-code**: optional probing routine inserted in the output header if configured.
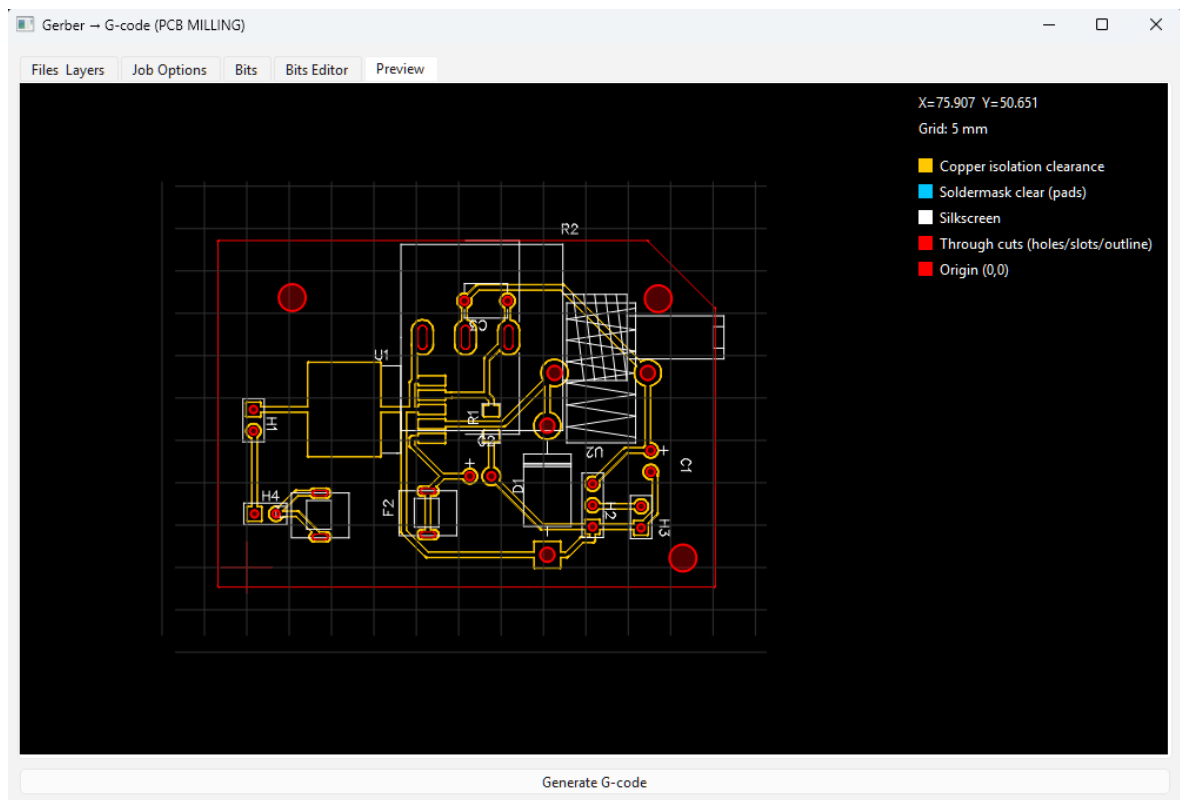
9. Preview tab

Figure 11 - Preview: validate orientation, origin, and toolpaths.

Preview normalizes coordinates using the top copper layer bounds so the minimum X/Y becomes (0,0). Set the CNC work zero (X0 Y0) to match this origin before running jobs.

10. Running the output on GRBL 1.1f

- Set units to millimeters (G21) and absolute mode (G90) if the sender does not do this automatically.

- Set the work origin to match the Preview origin.

- When the file pauses for tool changes (M0), change tools, re-zero Z if needed, then resume.

- Keep the PCB flat; V-bit isolation is sensitive to height variation.

11. Output files

- Combined output: a single file (commonly *all.nc*) with tool-change pauses.

- Separate output: one file per enabled operation.

- If an output directory is selected, generated .nc files are written/copied there.

12. Troubleshooting

- No operations detected: confirm a top copper and outline file exist, or load a complete ZIP export.

- Missing tool selection: create/select tools in Bits Editor for each enabled operation.

- Cuts appear shifted: re-check that the CNC work origin matches the Preview origin and that the machine is in mm units.

- Slot/large-hole warnings: the selected router tool may be too large for the feature; adjust tool diameter or settings.

13. Appendix: GRBL $ settings reference (common)

The settings below are common GRBL 1.1 parameters. Actual values depend on the specific machine mechanics and wiring.

13.1 Core motion and reporting

- **$0** Step pulse time (microseconds).
- **$1** Step idle delay (milliseconds).
- **$2** Step port invert mask.
- **$3** Direction port invert mask.
- **$4** Step enable invert.
- **$5** Limit pins invert.
- **$6** Probe pin invert.
- **$10** Status report mask (what GRBL reports back).
- **$11** Junction deviation (cornering tolerance).
- **$12** Arc tolerance.
- **$13** Report inches (0 = mm, 1 = inches).

13.2 Homing and limits

- **$20** Soft limits enable.
- **$21** Hard limits enable.
- **$22** Homing cycle enable.
- **$23** Homing direction invert mask.
- **$24** Homing locate feed rate.
- **$25** Homing search seek rate.
- **$26** Homing switch debounce (ms).
- **$27** Homing pull-off (mm).

13.3 Spindle and laser mode

- **$30** Max spindle speed (S value corresponding to full speed).

- **$31** Min spindle speed.

- **$32** Laser mode (0 = off, 1 = on).

13.4 Axis calibration and travel

- **$100**, **$101**, **$102** Steps/mm for X, Y, Z.

- **$110**, **$111**, **$112** Max rate (mm/min) for X, Y, Z.

- **$120**, **$121**, **$122** Acceleration (mm/s^2) for X, Y, Z.

- **$130**, **$131**, **$132** Max travel (mm) for X, Y, Z.