PROJECT ABSTRACT

CNC PCB Drilling Machine using GRBL

A 3-axis GRBL-based CNC machine that drills PCB through-holes from Gerber files — Arduino Uno with CNC shield, NEMA 17 steppers and an ER11 spindle — delivered as a built-to-order student project with the machine, firmware, workflow guide, report, PPT and viva kit.

01. Title

CNC PCB Drilling Machine using GRBL

02. Introduction / Background

Prototype PCBs sit between two poor options: chemical etching is messy and imprecise for fine holes, while fabrication houses add days and cost to every design iteration. A small CNC drilling machine closes that loop by reading the same Gerber and Excellon drill data a fab house uses and drilling through-holes on copper-clad board in the lab. GRBL — the open-source G-code interpreter for Arduino — makes such a machine a realistic student build: real-time motion planning, acceleration control and homing on commodity hardware. This project delivers that machine with the complete CAD-to-drill workflow.

03. Problem Statement

Electronics students need plated-style through-holes for component leads, but have no in-lab way to produce them accurately. Hand drilling wanders, etching cannot make holes at all, and outsourcing kills iteration speed. The project addresses this with a bench-top 3-axis drilling machine that turns drill files into accurately placed holes, while teaching G-code, stepper-motor control, machine calibration and the digital-fabrication toolchain.

04. Objectives

- Build a 3-axis gantry machine on 2020 aluminum extrusion with a design-target work area of ~200 x 150 x 60 mm.
- Implement GRBL v1.1 on an Arduino Uno + CNC Shield V3 driving three NEMA 17 steppers through A4988 drivers at 1/16 microstepping.
- Fit a 775 DC spindle with ER11 collet for 0.6–3.175 mm tungsten-carbide bits, with mechanical endstops and a homing cycle on all axes.
- Achieve a positioning-repeatability design target of ±0.1 mm, verified with motion tests during the build.

- Document the KiCad → Gerber/Excellon → FlatCAM → G-code-sender workflow so a student drills a real board in one sitting.
- Deliver the machine, firmware configuration, drawings, BOM, report, PPT and viva Q&A.

05. Proposed Methodology

The mechanical build starts with the 2020-extrusion frame, 8 mm linear rods with LM8UU bearings for X/Y and a lead screw for Z, followed by NEMA 17 mounting and spindle assembly. Electronics wiring connects the CNC shield, drivers, endstops, spindle supply and emergency stop. GRBL is flashed and configured through its $ parameters: steps/mm calibrated per axis, homing enabled, and acceleration tuned for the gantry mass. A test PCB layout is exported from KiCad, drill G-code is generated in FlatCAM with conservative feeds and peck depths, and the first boards are drilled, inspected for hole placement, and used to validate repeatability.

06. System Architecture / Working

The PCB is designed in KiCad and Gerber plus Excellon drill files are exported. FlatCAM converts the drill file to G-code with tool diameter, feed and peck settings. The board is clamped on the MDF spoil bed, axes are homed (GRBL $H), and work zero is set at the board corner. The G-code streams over USB from Universal Gcode Sender; GRBL plans trapezoidal acceleration and pulses the steppers through the A4988 drivers while the spindle spins the carbide bit and the Z axis pecks each hole to depth.

07. Hardware / Software Requirements

- Arduino Uno + CNC Shield V3; 3x NEMA 17 steppers (1.7 A/phase class); A4988 drivers
- 775 DC spindle motor (12–24 V) with ER11 collet; tungsten-carbide bits 0.6–1.2 mm
- 2020 aluminum-extrusion frame; 8 mm linear rods + LM8UU bearings; Z lead screw
- Mechanical endstops on X/Y/Z; panel emergency stop; 24 V DC supply
- KiCad (PCB design), FlatCAM (drill G-code), Universal Gcode Sender (streaming); GRBL v1.1 firmware

08. Expected Outcomes

An assembled, wired and motion-tested drilling machine that drills 0.6–1.2 mm through-holes from Gerber drill data, with homing, work coordinates and peck drilling demonstrated on real FR4. Repeatability is reported as the build's measured design-target verification, not a certified machine-tool spec. The student receives the full digital-fabrication workflow, usable for every future electronics prototype.

09. Applications

- In-lab PCB prototyping for electronics coursework and projects
- Teaching CNC fundamentals: G-code, homing, work coordinates, feeds and speeds
- Electronics-lab infrastructure: a shared drilling resource
- Foundation for isolation-milling and other subtractive extensions

10. Future Scope

- V-bit isolation milling for trace cutting (optional extension)
- Automatic tool-length probing and height mapping
- Enclosure with dust extraction for FR4 dust control
- Larger work area and ball-screw upgrade path
- Camera-assisted fiducial alignment for double-sided boards