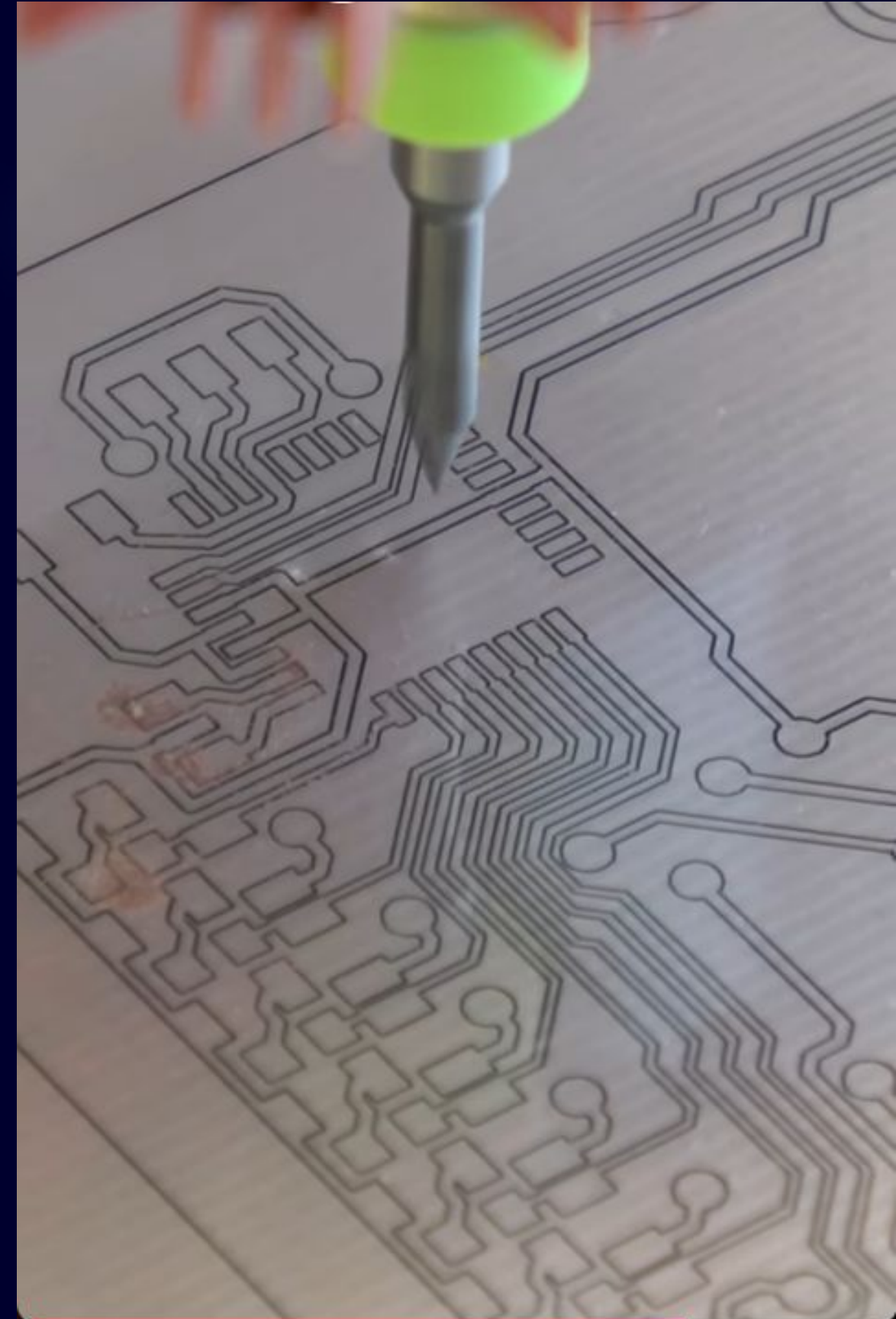


PCB Milling: From Design to Functional Circuit

Dive into the world of Printed Circuit Board (PCB) milling, a subtractive fabrication process that carves circuits from copper-clad boards. This method offers unparalleled **speed** for rapid prototyping, keeping development **cost-effective** for small batches. It's incredibly **accessible** for hobbyists and educational labs, providing the **flexibility** to quickly iterate designs and bring electronic ideas to life.



INTRODUCTION

The Essence of PCB Milling



Subtractive Process

Material is precisely removed from a copper-clad board to define the circuit.



Rapid Prototyping

Quickly iterate and test designs without waiting for external manufacturers.



In-House Fabrication

Perfect for FabLabs, research facilities, and educational institutions wanting autonomy.

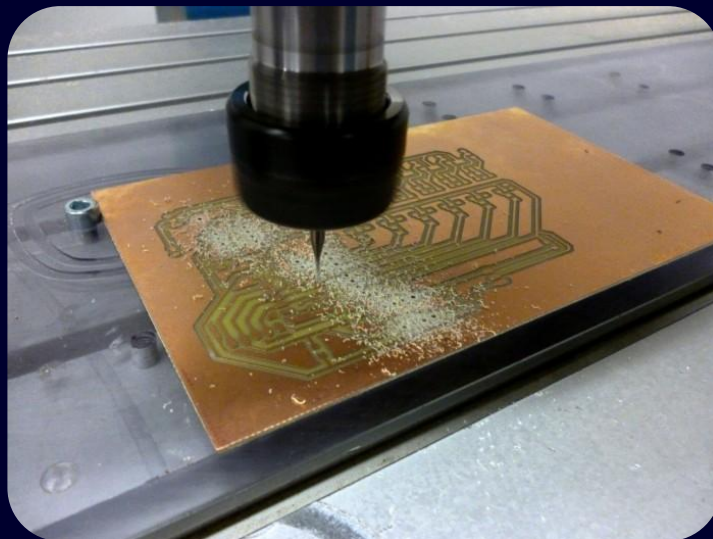
PCB milling trades the constraints of mass production for unparalleled **speed and flexibility**, empowering innovation right at your fingertips.

Choosing Your Fabrication Method

Understanding the different PCB fabrication methods is key to selecting the right approach for your project.

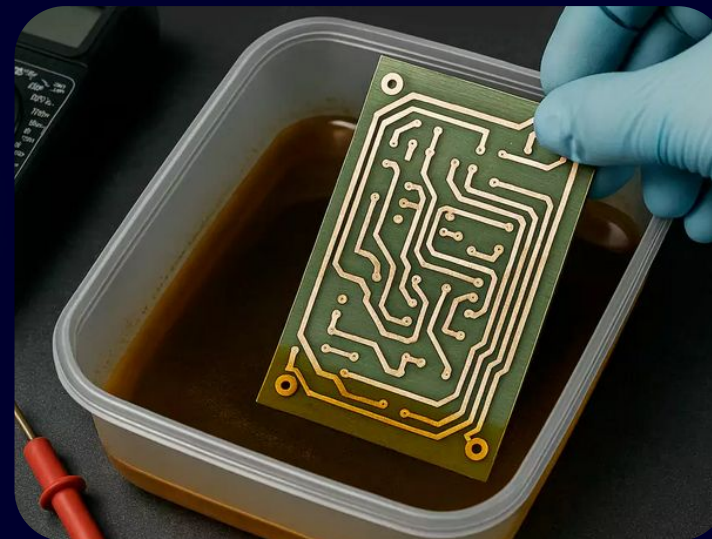
PCB Milling

Fast, chemical-free, best for prototypes and quick iterations. Limited by precision for very fine features.



Chemical Etching

Offers better surface quality and finer traces. Involves hazardous chemicals and disposal considerations.



Industrial Manufacturing

Highest precision, multi-layer capabilities, and cost-effective for large volumes. Long lead times and higher initial costs.



Ultimately, the choice depends on your project's **quantity, precision requirements, and time constraints.**

PCB Design for Milling Success

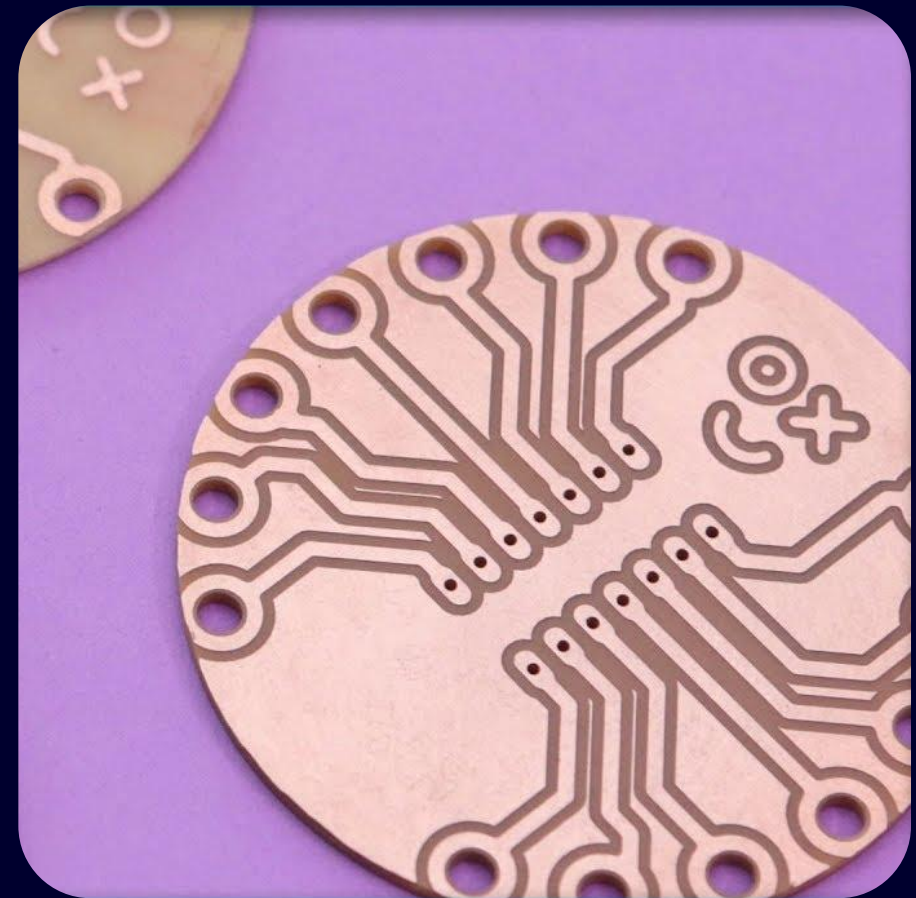
Designing with milling in mind ensures a smooth transition from digital concept to physical board. Consider the limitations and advantages of the process from the start.

Milling-friendly designs are typically **single-sided** to simplify the process.

Prioritize **through-hole or large SMD components** due to tool size limitations.

Key considerations include **tool size, trace spacing, and board thickness**.

Always **design for the machine**, not just the circuit – this is paramount for successful milling.



A well-planned design minimizes issues during fabrication and improves board reliability.

RULES

Understanding Traces, Clearance, Pads & Vias

1

Trace Width

Affects current carrying capacity and machinability. Wider traces are more robust.

2

Clearance

The isolation space between traces and copper areas. Crucial for preventing shorts.

3

Pads

Must be sufficiently large to ensure reliable soldering and structural integrity.

4

Vias

Often challenging to mill, consider replacing with **jumpers** for simpler prototypes.

For typical FabLab milling, aim for a minimum **trace width** of ≥ 0.4 mm and **clearance** of ≥ 0.4 mm to ensure machinability.

Design Rules for Manufacturability (DFM)

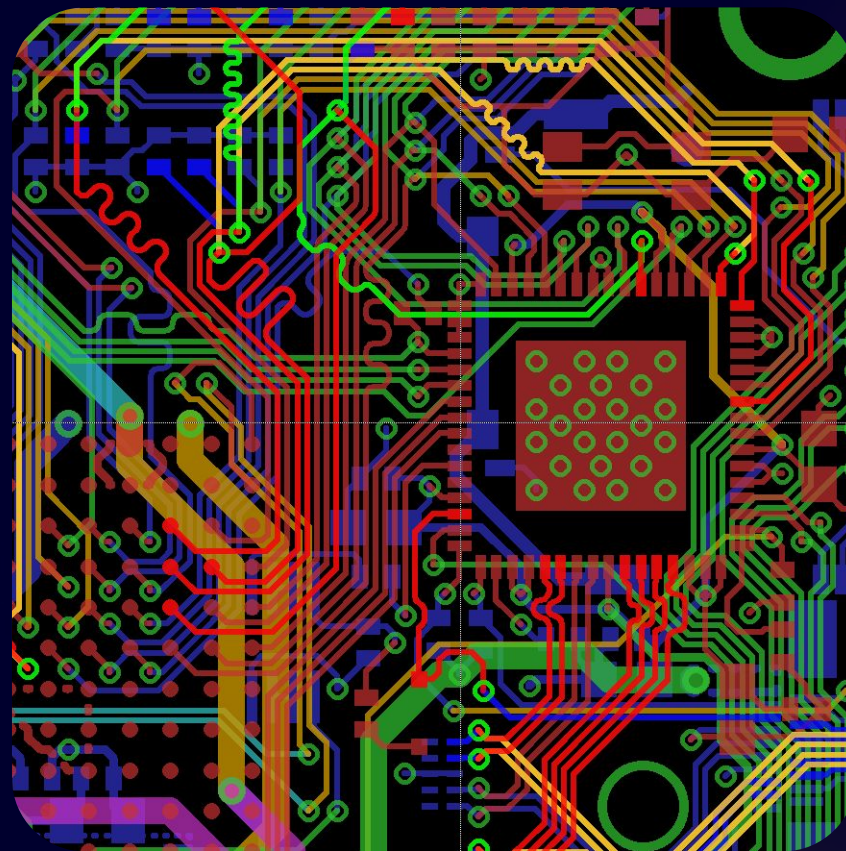
Applying DFM principles helps avoid common milling pitfalls and ensures a high-quality PCB.

Avoid These:

Very thin traces: Prone to breaking or lifting.

Sharp internal corners: Difficult for milling tools to replicate accurately.

Dense routing: Increases the risk of shorts and milling errors.

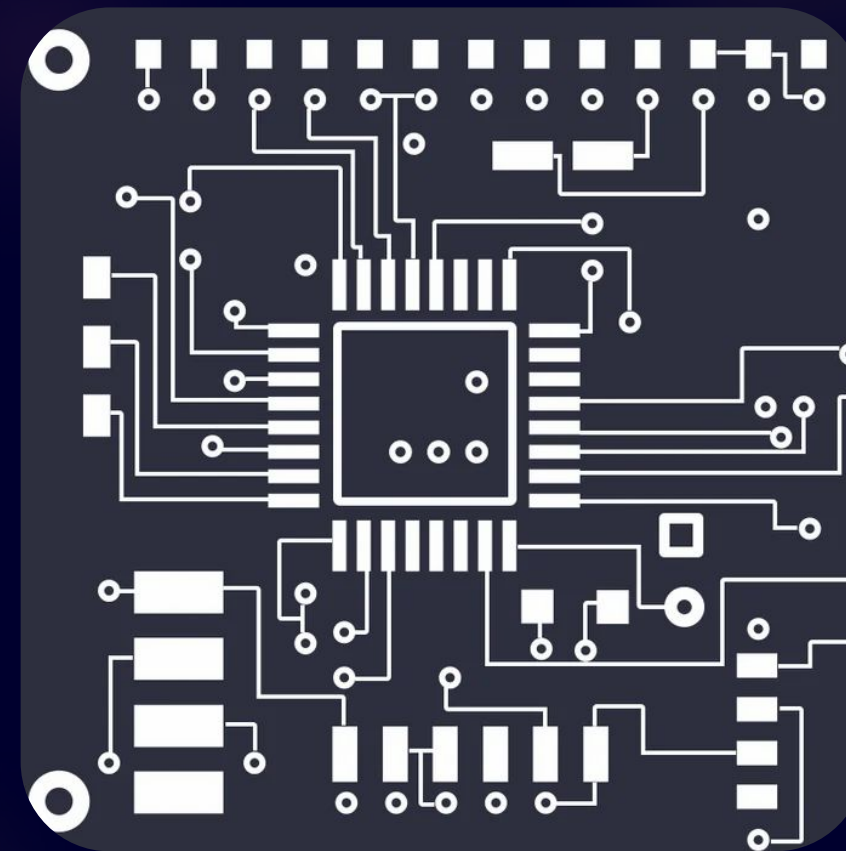


Prioritize These:

Larger pad sizes: Improves solderability and durability.

Increased clearance: Reduces the chance of accidental shorts.

Robust trace designs: Enhances the overall reliability of the circuit.



Remember the core DFM principle for milling: **If the bit can't reach it, don't design it.**

Machines, Tools & Calibration Essentials

The right equipment and meticulous calibration are the backbone of successful PCB milling.

Desktop CNC Mills

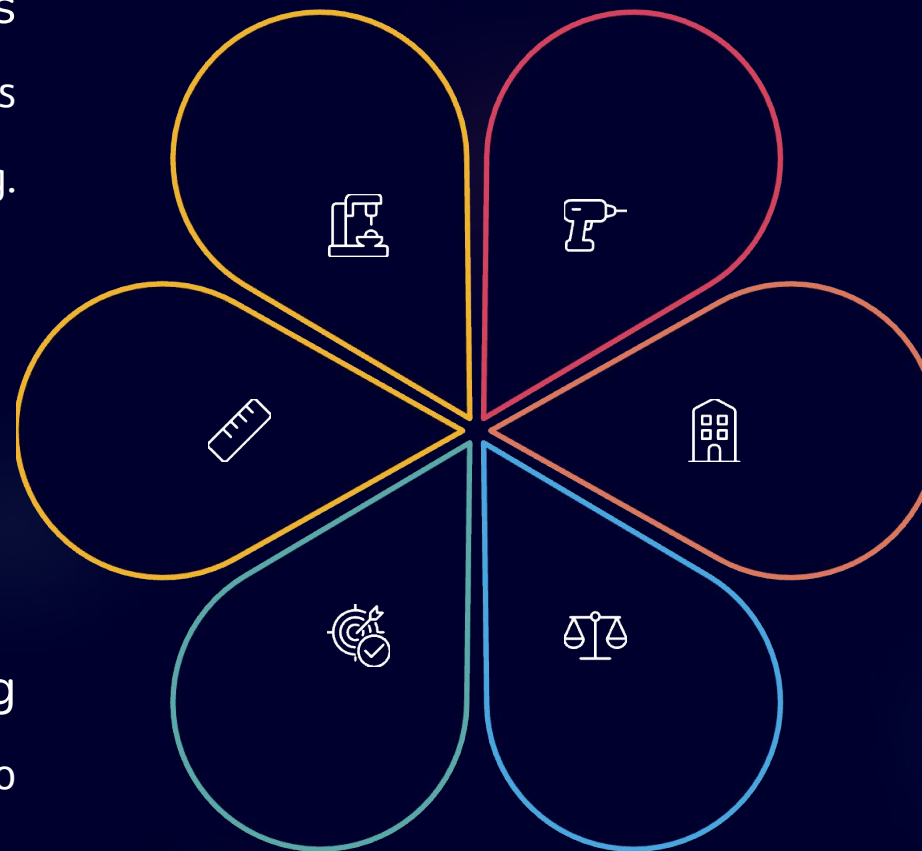
Commonly found in FabLabs, these machines offer precision and control for prototyping.

Board Flatness

Ensuring the PCB blank is perfectly flat prevents uneven milling and broken traces.

Tool Zeroing

Precisely setting the tool's starting point to ensure accurate milling.



V-Bits

Ideal for engraving and creating isolation traces on copper.

Flat End Mills

Used for clearing large copper areas and cutting board outlines.

Z-Height Leveling

Critical for consistent milling depth across the entire board surface.

Poor calibration is the enemy of successful milling, often leading to **broken traces and unusable boards**.

From Gerber to Toolpath: The Digital Bridge

The journey from your PCB design to a physical board involves crucial digital translation steps.



PCB Design Software

Create your circuit layout and export industry-standard Gerber files (traces, board outline, drills).



CAM Software

Convert Gerber files into machine-readable toolpaths. Define cut depth, tool diameter, and number of passes.



G-Code Generation

The CAM software generates G-code, the language that directs the CNC milling machine.



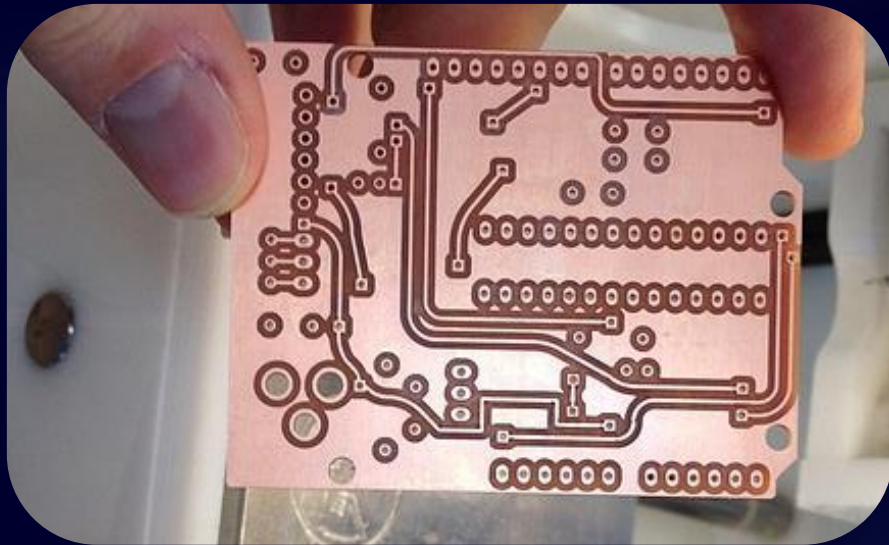
Machine Execution

The CNC mill executes the G-code, bringing your design to life on the copper board.

This sequence—**Design** → **Gerber** → **Toolpath** → **Machine**—is fundamental to the milling process.

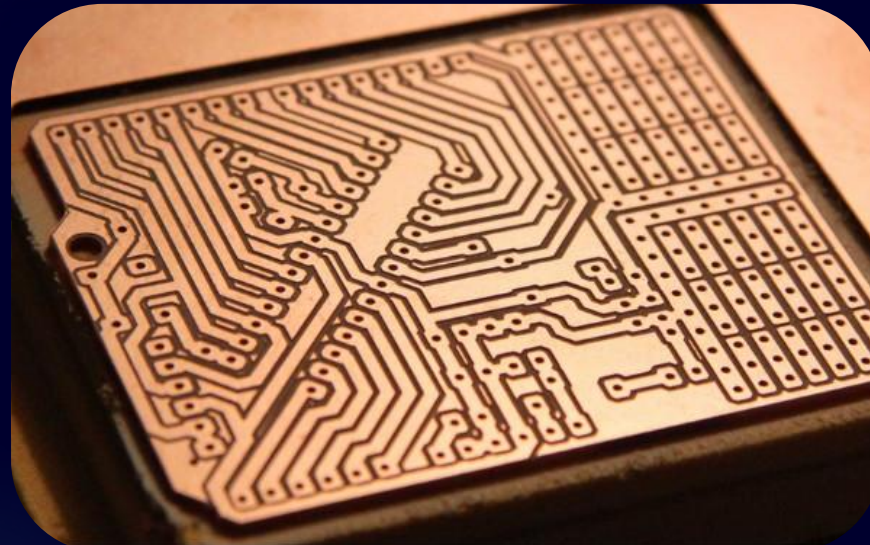
Milling, Drilling & Board Profiling

The physical fabrication involves three distinct stages, each requiring precision and proper sequencing.



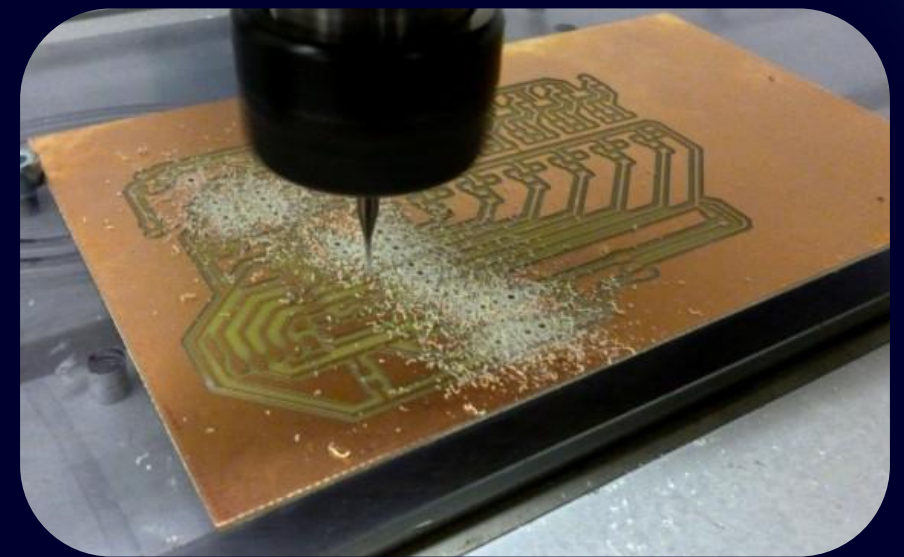
Trace Isolation

The milling tool removes copper around traces, creating the circuit paths.



Hole Drilling

Precise holes are drilled for component leads and vias.



Board Profiling

The final step, cutting the complete PCB outline from the stock material.

❏ Crucial Tip: Always mill traces and drill holes **before** cutting the board free to avoid misalignment.

POST-PROCESSING

Post-Processing, Soldering & Testing

Your PCB isn't finished until it works. These crucial final steps ensure functionality and reliability.

Clean & Inspect

Remove dust, burrs, and visually inspect traces for integrity and any unintended shorts.



Solder Components

Carefully attach all electronic components to the board.



Conduct Tests

Perform continuity checks, short circuit tests, and verify power rails.

Reality check: Most PCB failures are discovered **after** milling, making thorough testing indispensable.