

Introduction to CNC Milling

Introduction to CNC Milling

Computer Numerical Control (CNC) milling is a manufacturing process that uses computer-controlled rotary cutters to remove material from a workpiece and produce precise parts. Below is a concise, organized overview suitable for study or quick reference.

1. Basic Concepts

Term	Definition
CNC	Computer Numerical Control – a system that interprets G-code (or similar) to move machine axes automatically.
Milling	A machining operation where a rotating multi-tooth cutter removes material from a workpiece.
Workpiece	The raw material (metal, plastic, wood, etc.) that will be shaped.
Toolpath	The programmed trajectory of the cutter tip, defined in the CNC program.
Axes	Linear movements (X, Y, Z) and optionally rotary axes (A, B, C).
Spindle	The rotating shaft that holds the cutting tool; speed measured in RPM.

2. Machine Types

Machine	Typical Use	Key Features
Vertical Milling Machine (VMC)	General-purpose parts, prototyping	Spindle oriented vertically; easy chip evacuation.
Horizontal Milling Machine (HMC)	High-volume production, large parts	Spindle oriented horizontally; multiple cutters, better rigidity.
5-Axis Machining Center	Complex geometries, aerospace, molds	Simultaneous movement on X, Y, Z, plus two rotary axes (A, B or B, C).
Mini/Benchtop CNC	Education, small parts, hobbyists	Compact footprint, lower power, limited travel.

3. CNC Milling Workflow

Introduction to CNC Milling

1. **Design** – Create a 3D CAD model of the part.
 2. **CAM Programming** – Convert the CAD model into toolpaths using CAM software (e.g., Fusion 360, Mastercam).
 - Choose cutting tools (diameter, material, flute count).
 - Define machining operations (roughing, finishing, drilling, tapping).
 - Set parameters: feed rate, spindle speed, depth of cut, step?over.
 3. **Post?Processing** – Generate G?code (or other machine language) specific to the target CNC controller.
 4. **Machine Setup**
 - Install the workpiece (clamping, fixtures, vises).
 - Load the selected tool(s) into the tool magazine or spindle.
 - Zero the part (set the origin) using a probe or manual touch?off.
 5. **Dry Run / Simulation** – Verify toolpaths without material to catch errors.
 6. **Machining** – Run the program; monitor for tool wear, chip removal, and vibration.
 7. **Inspection** – Measure finished part (CMM, calipers, vision systems) to confirm tolerances.
 8. **Post?Processing** – Deburr, clean, and apply any required surface treatments.
-

4. Key Parameters

Parameter	Influence	Typical Units
Spindle Speed (RPM)	Determines cutting speed; too low ? poor surface finish, too high ? tool wear.	revolutions per minute
Feed Rate	Speed at which the cutter moves through material; affects productivity & tool life.	mm/min or inches/min
Depth of Cut (DOC)	Thickness of material removed per pass; impacts load on spindle & tool.	mm or inches

**Step?