

Introduction to CNC Lathe Operations

Introduction to CNC Lathe Operations

Designed as a quick-reference guide for students, technicians, and engineers.

1. What Is a CNC Lathe?

Feature	Description
CNC	Computer Numerical Control – the machine follows a programmed set of instructions (G-code/M-code).
Lathe	A rotating workpiece is shaped by stationary cutting tools.
CNC Lathe	Combines both: the spindle rotates the part automatically, while the tool turret moves in X (cross-slide) and Z (longitudinal) axes under computer control.

Typical Applications

- Turning (external cylindrical surfaces)
 - Facing (end-wise machining)
 - Boring & drilling (internal features)
 - Thread cutting (metric & imperial)
 - Grooving, knurling, taper turning, and contouring
-

2. Main Components & Axes

Component	Function
Spindle	Holds and rotates the workpiece (speed selectable).
Chuck / Collet	Grips the workpiece; can be 3-jaw, 4-jaw, or collet for precision.
Turret	Holds multiple tools; rotates to index the required tool.
Cross-Slide (X-axis)	Moves the tool radially (perpendicular to spindle axis).
Longitudinal Slide (Z-axis)	Moves the tool parallel to the spindle axis.
Tool Post / Sub-Turret	Provides additional tool offsets or secondary operations.
Control Panel	Displays the program, status, and allows manual overrides.
Coolant System	Delivers cutting fluid to reduce temperature and chip adhesion.

Introduction to CNC Lathe Operations

Axes Overview

Axis	Direction	Typical Use
X	Radial (toward/away from center)	Cutting depth, diameter control.
Z	Longitudinal (along spindle)	Length, facing, taper.
C (optional)	Rotational (turning of spindle)	Threading, helical cuts.

3. CNC Lathe Programming Basics

3.1. G?Code Structure

```
O[program number]      (Program start)
N[line number] G[code] X[value] Z[value] F[feed] S[spindle] M[code]
...
M30                    (Program end)
```

3.2. Common G?Codes (ISO standard)

Code	Meaning	Typical Use
G00	Rapid positioning	Move tool quickly without cutting.
G01	Linear interpolation (feed)	Straight?line cuts at feed rate F .
G02	Clockwise circular interpolation	Cutting arcs, threads (C?axis).
G03	Counter?clockwise circular interpolation	Same as G02, opposite direction.
G20	Inch mode	Units = inches.
G21	Metric mode	Units = millimeters.
G40	Cancel cutter radius compensation (CRC).	
G41	Left cutter radius compensation.	
G42	Right cutter radius compensation.	
G70	Finishing cycle (light cut).	
G71	Rough turning cycle (multiple passes).	
G73	High?speed drilling cycle.	
G76	Thread cutting cycle (multiple passes).	
G90	Absolute positioning.	
**G91		