

Overview of CNC Lathe Operations

Overview of CNC Lathe Operations

Compact, easy-to-read study notes

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	Rotating workpiece (spindle) while a cutting tool moves linearly to shape the material.
Primary Use	Turning operations: external diameters, internal diameters, threads, tapers, grooves, and face-machining.

2. Main Components

Component	Function
Spindle	Holds and rotates the workpiece (speed range 30–6000 rpm typical).
Tool Turret	Holds 8–12 tools; rotates to bring the required tool into the cutting position.
Carriage	Moves the turret (X-axis) and cross-slide (Z-axis).
Chuck/Collet	Secures the workpiece; may be 3-jaw, 4-jaw, or collet type.
Control Panel	Displays the program, status, and allows manual overrides.
Coolant System	Delivers cutting fluid to reduce heat and improve surface finish.

3. Axis System (Standard CNC Lathe)

Axis	Direction	Typical Motion
X	Radial (perpendicular to spindle axis)	Tool moves toward/away from workpiece – controls diameter.
Z	Longitudinal (parallel to spindle axis)	Tool moves along the length of the part – controls length & facing.

Overview of CNC Lathe Operations

Axis	Direction	Typical Motion
C	Rotational (spindle)	Controls spindle speed and rotation for threading or indexing.
U / V	Optional secondary linear axes (rare on basic lathes).	

4. Basic CNC Lathe Operations

Operation	Goal	Typical G?code(s)
Facing	Create a flat end surface (face) at the workpiece tip.	G01 (linear feed) + X?axis movement to the desired diameter.
Turning (Straight Turning)	Reduce diameter uniformly along a length.	G01 X?axis feed while Z moves.
Grooving	Cut a narrow recessed slot.	G01 + X & Z moves; often use a smaller tool radius.
Parting (Cut?off)	Separate the finished part from the bar.	G01 X?axis cut to the required depth, then retract.
Thread Cutting	Produce internal or external threads.	G76 (threading cycle) or G32 (single?pass thread).
Taper Turning	Produce a conical shape.	G01 with simultaneous X & Z motion (or G41/G42 for tool radius compensation).
Drilling/Boring	Create holes on the workpiece axis.	G81 (drilling cycle) or G85 (boring cycle) with Z?axis movement.
Chamfering	Add a beveled edge.	G01 with a small X offset at the end of a cut.

5. Programming Essentials

1. Program Structure

```
%  
O#### (program number)  
(Optional comment)  
G20 (inches) or G21 (mm) ; unit selection
```

Overview of CNC Lathe Operations

G40 (cancel cutter radius comp.)

G90 (absolute programming) ; or G91 for incremental

G