

G-Code Cheat Sheet provided by OpenAI's ChatGPT 4o

Below is an overview of G-code commands for CNC machines.

Basic G-Code Commands

Movement Commands

- **G00** - Rapid Positioning (Fastest movement to a specified location)
- **G01** - Linear Interpolation (Controlled movement at a set feed rate)
- **G02** - Circular Interpolation Clockwise (CW Arc)
- **G03** - Circular Interpolation Counter-Clockwise (CCW Arc)
- **G04** - Dwell (Pause movement for a specified time)

Unit and Coordinate System Setup

- **G20** - Use Inches for Units
- **G21** - Use Millimeters for Units
- **G90** - Absolute Positioning (All coordinates are from the origin)
- **G91** - Relative Positioning (All coordinates are relative to the last position)
- **G92** - Set Position (Assigns the current position as a specified coordinate)

Spindle and Feed Rate Control

- **M03** - Spindle ON (Clockwise)
- **M04** - Spindle ON (Counter-Clockwise)
- **M05** - Spindle OFF
- **Fxxx** - Set Feed Rate (e.g., F100 sets feed to 100 units/min)

Tool and Program Control

- **M00** - Program Stop (Pause until restarted)
- **M01** - Optional Stop (Stops only if optional stop is enabled)
- **M02** - Program End (Stops execution and rewinds program)
- **M30** - Program Reset (Stops execution and resets position)
- **Txx** - Tool Change (e.g., T1 changes to tool 1)
- **M06** - Execute Tool Change

Coolant and Other Functions

- **M07** - Mist Coolant ON
- **M08** - Flood Coolant ON
- **M09** - Coolant OFF
- **M10** - Clamp Workpiece
- **M11** - Release Workpiece

Commonly Used Parameters

- **X, Y, Z** - Position Coordinates
- **I, J, K** - Arc Center Offsets (Used with G02/G03)
- **Sxxx** - Spindle Speed (e.g., S1500 sets spindle to 1500 RPM)
- **R** - Radius (Used for arcs and canned cycles)

Canned Cycles (Drilling & Tapping)

- **G81** - Simple Drilling Cycle
- **G82** - Drilling Cycle with Dwell
- **G83** - Peck Drilling Cycle
- **G84** - Tapping Cycle
- **G85** - Boring Cycle

Additional Notes

- Always ensure units are correctly set before running a program.
- Rapid movements (G00) do not consider feed rate and can be dangerous if misused.
- Circular interpolation (G02/G03) requires a center offset with I, J, K values.
- Absolute (G90) vs. Relative (G91) positioning significantly affects tool movement.

This cheat sheet serves as a quick reference for fundamental G-Code commands. For a more in-depth understanding, refer to CNC machine documentation or software-specific guidelines.