

G & M-code cheat sheet

G-Code	Meaning
G00	Rapid Positioning
G01	Linear Interpolation
G02	Clockwise Circular/Helical Interpolation
G03	Counterclockwise Circular/Helical Interpolation
G04	Dwell
G10	Coordinate System Origin Setting
G12	Clockwise Circular Pocket
G13	Counterclockwise Circular Pocket
G15/16	Polar Coordinate Moves in G00 & G01
G17	XY Plane Select
G18	XZ Plane Select
G19	YZ Plane Select
G20	Inch Unit
G21	Millimeter Unit
G28	Return Home
G28.1	Reference Axes
G30	Return Home
G31	Straight Probe
G40	Cancel Cutter Radius Compensation
G41	Start Cutter Radius Compensation (left)
G42	Start Cutter Radius Compensation (right)
G43	Apply Tool Length Offset (plus)
G49	Cancel Tool Length Offset
G50	Reset All Scale Factors to 1.0

G & M-code cheat sheet

G-Code	Meaning
G51	Set Axis Data Input Scale Factors
G52	Temporary Coordinate System Offsets
G53	Move in Absolute Machine Coordinate System
G54	Use Fixture Offset 1
G55	Use Fixture Offset 2
G56	Use Fixture Offset 3
G57	Use Fixture Offset 4
G58	Use Fixture Offset 5
G59	Use Fixture Offset 6 – Use General Fixture #
G61	Exact Stop Mode
G64	Constant Velocity Mode
G68	Rotate Program Coordinate System
G70/71	Inch/Millimeter Unit
G73	Canned Cycle – Peck Drilling
G80	Cancel Motion Mode (Including canned cycles)
G81	Canned Cycle – Drilling
G82	Canned Cycle – Drilling with Dwell
G83	Canned Cycle – Peck Drilling
G85/86/88/89	Canned Cycle – Boring
G90	Absolute Distance Mode
G91	Incremental Distance Mode
G92	Offset Coordinates & Set Parameters
G92.X	Cancel G92, Etc.
G93	Inverse Time Feed Mode

G & M-code cheat sheet

G-Code	Meaning
G94	Feed per Minute Mode
G95	Feed per Revolution Mode
G98	Initial Level Return after Canned Cycles
G99	R-point Level Return after Canned Cycles

M-Code	Meaning
M00	Program Stop
M01	Optional Program Stop
M02	Program End
M03	Torch On/Rotate Spindle Clockwise
M04	Rotate Spindle Counter-Clockwise
M05	Torch Off/Stop Spindle Rotation
M06	Tool Change (by 2 macros)
M07	Mist Coolant On
M08	Flood Coolant On
M09	All Coolant Off
M30	Program End & Rewind
M47	Repeat Program from 1 st Line
M48	Enable Speed & Feed Override
M49	Disable Speed & Feed Override
M98	Call Subroutine
M99	Return from Subroutine/Repeat