Code	Meaning	Letter	Meaning
G00	Positioning (Rapid Traverse)	A	Rotary indexing axis around the "X" axis
G01	Linear Interpolation (Feed)	B	Rotary indexing axis around the "Y" axis
G02	Circular Interpolation CW	C	Rotary indexing axis around the "Z" axis
G03	Circular Interpolation CCW	D	Cutter Radius / Diameter offset number
G04	Dwell	E	Feedrate in inches per revolution (Lathe)
G17	X,Y Plane of Interpolation	F	Feedrate in inches per minute (can be used on lathes)
G20	Input in Inch	G	Preparatory commands
G21	Input in MM	H	Offset Number (Mill - tool length)(Lathe - Position offset)
G28	Return to Machine zero (reference point)	I	Arc center location in the "X" axis
G40	Cutter Compensation cancel	J	Arc center location in the "Y" axis
G41	Cutter Compensation left	K	Arc center location in the "Z" axis
G42	Cutter Compensation right	L	Fixed Cycle repetion count / subprogram repetition count
G43	Tool length compensation +	M	Miscellaneous function
G54-G59	Set Local Coordinate Systems (Datum Shifts)	N	Block / Sequence number
G80	Canned Cycle cancel		
G81	Spot Drilling Cycle (no dwell)	O	Program Number
G82	Drill/Counterbore (with dwell)	P	Subprogram / Macro Number call
G83	Peck Drilling Cycle	P	Dwell time in milliseconds
G73	Peck Drilling Cycle (High Speed)	P	Block number in main program when used with M99
G84	Tapping Cycle (RH)	Q	Depth of peck in fixed cycles (G73and G83)
G85	Bore	Q	Shift amount in fixed cycle (G76 and G87)
G90	Absolute Programming	R	Retract point in fixed cycles
G91	Incremental Programming	R	Arc Radius designation
G92	Set Program Zero	S	Spindle speed in RPMs
M00	Program stop	T	Tool function
M01	Optional stop	U	Incremental move in "X" axis
M02	End of Program	V	Incremental move in "Y" axis
M03	Start Spindle (Clockwise)	W	Incremental move in "Z" axis
M04	Start Spindle (Counterclockwise)	X	"X" axis coordinate value designation
M05	Spindle off	Y	"Y" axis coordinate value designation
M06	Tool change	Z	"Z" axis coordinate value designation
M08	Coolant on		
M09	Coolant off		
M30	Program reset/tape rewind		
M98	Sub-Program call		
M99	Return to previous program		