

G Code	Description	Example
F	Feed rate is followed by a feed speed. Inches or mm per minute.	F180
V	Set or use a variable. Specify V0 through V100. Use V0 through V100 .	V59=2.34 G01 XV59
X	Location or distance to move in the x axis.	X3
Y	Location or distance to move in the y axis.	Y3
Z	Location or distance to move in the z axis.	Z3
W	Location or distance to move in the auxiliary axis this character may not be "W", it could also be "A" or "B".	W3
G0	Fast move. Rapid positioning command (no cutting)	G00 X0 Y0 Z0
G1	Linear move. Straight line cutting at feed rate.	G01
G2	Clockwise rotation cutting motion.	G02
G3	Counterclockwise rotation cutting motion.	G03
G4	Dwell followed by a P parameter with a dwell time in milliseconds. dwells for 2 seconds	G4 P2000
G17	XY plane for arcs assigns circular interpolation.	G17
G18	ZX plane for arcs assigns circular interpolation.	G18
G19	YZ plane for arcs assigns circular interpolation.	G19
G20	Inch units	G20
G74	Inch units	G74
G21	Metric units	G21
G75	Metric units	G75
G22	Do subroutine followed by a P parameter with a subroutine number eg. "G22 P20" up to 100 subroutines can be specified in a program. Each subroutine is defined after the M30 statement and is signified by the '\$' character. M00 is not supported within any subroutine and if called, the subroutine will be terminated. Each subroutine is terminated by M02.	G22 P20 V1=2 V2=1 G22 P20 V1=5 V2=3 M30 \$20 XV1 YV2 X0 Y0 M2
G43	New tool followed by a P parameter with a tool number (tool length compensation)	G43 P11
G49	Cancel tool length compensation	G49
G50	Cancel scaling	G50
G51	Scaling followed by a P parameter with a scale factor and also followed by X, Y, and Z parameters determining scaling point	G51 P1.5 X3 Y3 Z2
G53	Machine coordinates - system setting (fixture home position) used after G54-G59 to cancel offsets. Must be preceded by G00 or G01.	
G54	Offset 2 G54-G59:	
G55	Offset 3 Work coordinate system offsets (fixture offsets)	
G56	Offset 4 Home position for different (fixture offsets)	
G57	Offset 5 Home position for different parts.	
G58	Offset 6	
G59	Offset 7	
G60	Constant contouring OFF	
G64	Constant contouring ON	
G70	Independant auxiliary axis	
G71	Auxiliary axis follow X-axis	
G72	Auxiliary axis follow Y-axis	
G73	Auxiliary axis follow Z-axis	
G80	Cancel drill cycle	
G81	Single pass drill cycle on. Up position specified by R parameter. Drill position depth specified by Z parameter. 1st position specified before G81. where parameters are in absolute coords	G81 R1 Z-.5
G83	Multiple pass drill cycle on. Up position specified by R parameter. Drill position depth specified by Z parameter. Max depth/pass specified by Q parameter where parameters are in absolute coords. Except Q (pass depth).	G83 R1 Z-.5 Q-.25
G90	Absolute coordinates positioning mode	G90
G91	Relative coordinates (incremental) positioning mode	G91
G92	Reset machine coordinates to the coodinates specified by the following X, Y, and Z parameters -only works while in G53.	G92 X0 Y0 Z0
G95	Quit executing current file and begin executing next file specified (can not use within a subroutine)	G95 #c:\path\to\file\file.ext
L	(Can not use within a subroutine) Loop a line of Gcode looping parameters specified by a number eg: "L100" Executes the line of code it's in 100 times.	L

M Code	Description
M02	End of program
M03	User defined output 1 on
M05	User defined output 1 off
M07	User defined output 2 on
M08	User defined output 3 on
M09	User defined output 2 & 3 off
M10	User defined output 4 on
M11	User defined output 4 off
M06	Go to tool change position
M00	Cycle stop (program stop)
M30	Cycle stop and rewind (program end)
M99	Cycle stop and rewind and cycle begin
M100	Rotate +90 deg. about the X-axis
M101	Rotate -90 deg. about the X-axis
M102	Rotate +90 deg. about the Y-axis
M103	Rotate -90 deg. about the Y-axis
M104	Rotate +90 deg. about the Z-axis (X and Y swapped) (rotate program)
M105	Rotate -90 deg. about the Z-axis (X and Y swapped) (un-rotate program)