

Practical No. 2

Aim:- Comparative study of international standard codes G-Codes and M-codes for major operations

Preparatory function (G-Codes).

Code	Description	Milling (M)	Turning (T)	Corollary info
G00	Rapid positioning	M	T	
G01	Linear interpolation	M	T	
G02	Circular interpolation, clockwise	M	T	.
G03	Circular interpolation, counter clockwise	M	T	
G04	Dwell	M	T	Takes an address for dwell period (may be X , U , or P). The dwell period is specified in the controller's parameter, typically milliseconds.
G17	XY plane selection	M		
G18	ZX plane selection	M	T	
G19	YZ plane selection	M		
G20	Programming in inches	M	T	
G21	Programming in millimetres (mm)	M	T	
G28	Return to home position (machine zero, aka machine reference point)	M	T	Takes X Y Z addresses which define the intermediate point that the tool tip will pass through on its way home to machine zero. They are in terms of part zero (aka program zero), NOT machine zero.
G40	Tool radius compensation off	M	T	Cancels G41 or G42.
G41	Tool radius compensation left	M	T	Milling: Given right hand-helix cutter and M03 spindle direction, G41 corresponds to climb milling (down milling) . Takes an address (D or H) that calls an offset register value for radius.

G42	Tool radius compensation right	M	T	Similar corollary info as for G41. Given right hand-helix cutter and M03 spindle direction, G42 corresponds to conventional milling (up milling) . See also the comments for G41 .
G43	Tool height offset compensation negative	M		Takes an address, usually H, to call the tool length offset register value. The value is <i>negative</i> because it will be <i>added</i> to the gauge line position. G43 is the commonly used version (vs. G44).
G44	Tool height offset compensation positive	M		Takes an address, usually H, to call the tool length offset register value. The value is <i>positive</i> because it will be <i>subtracted</i> from the gauge line position. G44 is the seldom-used version (vs. G43).
G49	Tool length offset compensation cancel	M		Cancels G43 or G44 .
G50	Scaling function cancel	M		
G52	Local coordinate system (LCS)	M		Temporarily shifts program zero to a new location. This simplifies programming in some cases.
G53	Machine coordinate system	M	T	
G54 to G59	Work coordinate systems (WCSs)	M	T	
G54.1 P1 to P48	Extended work coordinate systems	M	T	Up to 48 more WCSs besides the 6 provided as standard by G54 to G59.
G70	Fixed cycle, multiple repetitive cycle, for finishing (including contours)		T	

G71	Fixed cycle, multiple repetitive cycle, for roughing (Z-axis emphasis)		T	
G72	Fixed cycle, multiple repetitive cycle, for roughing (X-axis emphasis)		T	
G73	Fixed cycle, multiple repetitive cycle, for roughing, with pattern repetition		T	
G73	Peck drilling cycle for milling – high-speed (NO full retraction from pecks)	M		Retracts only as far as a clearance increment (system parameter). For when chip breaking is the main concern, but chip clogging of flutes is not.
G74	Peck drilling cycle for turning		T	
G74	Tapping cycle for milling, left-hand thread, M04 spindle direction	M		
G75	Peck grooving cycle for turning		T	
G76	Fine boring cycle for milling	M		
G76	Threading cycle for turning, multiple repetitive cycle		T	
G80	Cancel canned cycle	M	T	Milling: Cancels all cycles such as G73, G83, G81, and G86 etc. Z-axis returns either to Z-initial level or R-level, as programmed (G98 or G99, respectively).
G81	Simple drilling cycle	M		No dwell built in

G82	Drilling cycle with dwell	M		Dwells at hole bottom (Z-depth) for the number of milliseconds specified by the P address. Good for when hole bottom finish matters.
G83	Peck drilling cycle (full retraction from pecks)	M		Returns to R-level after each peck. Good for clearing flutes of chips.
G84	Tapping cycle, right-hand thread, M03 spindle direction	M		
G85	Reaming Cycle	M		
G86	Boring Cycle	M		
G90	Absolute programming	M	T	Positioning defined with reference to part zero...
G91	Incremental programming	M	T	Positioning defined with reference to previous position.
G92	Threading cycle, simple cycle		T	
G94	Feedrate per minute	M	T	
G95	Feedrate per revolution	M	T	
G96	Constant surface speed (CSS)		T	Varies spindle speed automatically to achieve a constant surface speed. See speeds and feeds . Takes an S address integer, which is interpreted as sfm in G20 mode or as m/min in G21 mode.
G97	Constant spindle speed	M	T	Takes an S address integer, which is interpreted as rev/min (rpm). The default speed mode per system parameter if no mode is programmed.
G98	Return to initial Z level in canned cycle	M		

G98	Feedrate per minute (group type A)		T	Feedrate per minute is G94 on group type B.
G99	Return to R level in canned cycle	M		
G99	Feedrate per revolution (group type A)		T	Feedrate per revolution is G95 on group type

Miscellaneous functions

M Codes are instructions describing machine functions such as calling the tool, spindle Rotation, coolant on, door close/open etc.

Code	Description	Milling (M)	Turning (T)	Corollary info
M00	Compulsory stop	M	T	Non-optional—machine will always stop upon reaching M00 in the program execution.
M01	Optional stop	M	T	Machine will only stop at M01 if operator has pushed the optional stop button.
M02	End of program	M	T	No return to program top; may or may not reset register values.
M03	Spindle on (clockwise rotation)	M	T	The speed of the spindle is determined by the address S.
M04	Spindle on (counter clockwise rotation)	M	T	See comment above at M03.
M05	Spindle stop	M	T	
M06	Automatic tool change (ATC)	M	T	
M07	Coolant on	M	T	
M08	Coolant on (flood)	M	T	
M09	Coolant off	M	T	

M19	Spindle orientation	M	T	Spindle orientation
M21	Mirror, X-axis	M		
M21	Tailstock forward		T	
M22	Mirror, Y-axis	M		
M22	Tailstock backward		T	
M23	Mirror OFF	M		
M30	End of program with return to program top and Rewind	M	T	
M98	Subprogram call	M	T	Takes an address P to specify which subprogram to call, for example, “M98 P8979” calls subprogram O8979.
M99	Subprogram end	M	T	