G-code and M-code cheat sheet (Fanuc milling)

Meanings are typical for Fanuc-style milling controls. Other controls and machine builders may differ – follow the machine documentation.

G-codes

CODE	MEANING
G00	Rapid traverse
G01	Linear interpolation (feed)
G02	Circular interpolation clockwise
G03	Circular interpolation counter-clockwise
G04	Dwell
G17 / G18 / G19	Plane XY / ZX / YZ
G20 / G21	Inch / metric input
G28	Return to reference point
G40	Cutter compensation cancel
G41 / G42	Cutter compensation left / right
G43 / G49	Tool length offset / cancel
G53	Machine coordinate move
G54–G59	Work coordinate systems
G73	High-speed peck drilling
G74 / G84	Left-hand / right-hand tapping
G76	Fine boring
G80	Canned cycle cancel
G81 / G82	Drilling / drilling with dwell
G83	Deep-hole peck drilling
G85–G89	Boring cycles
G90 / G91	Absolute / incremental
G94 / G95	Feed per minute / per revolution
G96 / G97	Constant surface speed / constant RPM
G98 / G99	Return to initial level / R plane

M-codes

CODE	MEANING
M00	Program stop
M01	Optional stop
M02	End of program
M03 / M04	Spindle clockwise / counter-clockwise
M05	Spindle stop

CODE	MEANING
M06	Tool change
M08 / M09	Coolant on / off
M19	Spindle orientation
M29	Rigid tapping
M30	End of program and rewind
M98 / M99	Subprogram call / return

M19, M29 and some cycle meanings depend on the machine builder.

© CNC Pasaulis · cncpasaulis.lt/en/charts/g-code-m-code-cheat-sheet/