



**G-COMMANDS FOR MILLING**

<b>G CODE MODE</b>	<b>EFFECT</b>
Modal	Remain in effect for all blocks unless replaced by another modal G code
Non-Modal	Only effect the block in which it appears

<b>MODE</b>	<b>COMMAND</b>	<b>FUNCTION</b>
None-Modal	G04	Dwell
None-Modal	G09	Exact stop
None-Modal	G10	Data setting
None-Modal	G11	Data setting OFF
None-Modal	G28	Approach reference point
None-Modal	G52	Local coordinate system
None-Modal	G53	Machine coordinate system
Modal	G92	Coordinate system setting
Modal	G00	Positioning (Rapid Traverse)
Modal	G01	Linear interpolation
Modal	G02	Circular interpolation clockwise
Modal	G03	Circular interpolation counterclockwise
Modal	G17	Plane selection XY
Modal	G18	Plane selection ZX
Modal	G19	Plane selection YZ
Modal	G90	Absolute programming
Modal	G91	Incremental programming
Modal	G94	Feed per minute
Modal	G95	Feed per revolution
Modal	G20	Measuring in inches
Modal	G21	Measuring in millimeter
Modal	G40	Cancel cutter radius compensation
Modal	G41	Cutter radius compensation left
Modal	G42	Cutter radius compensation right
Modal	G43	Tool length compensation positive
Modal	G44	Tool length compensation negative
Modal	G49	Cancel tool length compensation
Modal	G73	Chip break drilling cycle
Modal	G74	Left tapping cycle
Modal	G76	Fine drilling cycle
Modal	G80	Cancel drilling cycles
Modal	G81	Drilling cycle
Modal	G82	Drilling cycle with dwell
Modal	G83	Withdrawing drilling cycle
Modal	G84	Tapping cycle
Modal	G85	Reaming cycle
Modal	G86	Drilling cycle with spindle stop
Modal	G87	Back pocket drilling cycle
Modal	G88	Drilling cycle with program stop
Modal	G89	Reaming cycle with dwell
Modal	G98	Retraction to starting plane
Modal	G99	Retraction to withdrawal plane
Modal	G50	Cancel scale factor
Modal	G51	Scale factor
Modal	G97	Revelations per minute
None-Modal	G54-G59	Zero offset 1-6
Modal	G61	Exact stop mode
Modal	G63	Thread cutting mode ON
Modal	G64	Cutting mode
Modal	G68	Coordinate system rotation ON
Modal	G69	Coordinate system rotation OFF
Modal	G15	End polar coordinate interpolation
Modal	G16	Begin polar coordinate interpolation



**M-COMMANDS FOR MILLING**

<b>COMMAND</b>	<b>FUNCTION</b>
M0	Programmed stop
M1	Programmed stop, conditional
M2	Program end
M3	Spindle ON clockwise
M4	Spindle ON counterclockwise
M5	Main spindle OFF
M6	Tool change
M8	Coolant ON
M09	Coolant OFF
M10	Lock dividing head
M11	Unlock dividing head
M19	Oriented spindle stop
M25	Release clamping device
M26	Close clamping device
M30	Program end
M71	Puff blowing ON
M72	Puff blowing OFF
M98	Subroutine call
M99	Subroutine end, jump command

**NOTE:** In **FANUC 21** controller only one M-code per block is allowed. If two or more M-codes are used in the same block of code only one will be executed and the others will be ignored.