CNC PROGRAMMING

List of G- and M-codes with descriptions.

The most common G-code commands and M functions in one place: tool movements, zeros, compensation, canned cycles and machine functions. Descriptions follow ISO 6983 / DIN 66025 and the typical Fanuc dialect.

01 BLOCK STRUCTURE

N30	G01	X50.0	Y20.0	F200
N	G	X	Y	F

Block = one program line. Word = address (letter) + value.
Most G-codes are modal: they stay active until cancelled.

N	block number
G	motion type, mode
X Y Z	axis coordinates
F	feed
S	spindle speed
T	tool number
M	machine function

02 MOTION AND MODES

CODE	FUNCTION
G00	rapid traverse
G01	linear move at feed
G02 / G03	arc CW / CCW
G04	dwell
G09	exact stop
G17 / G18 / G19	plane XY / ZX / YZ
G20 / G21	inches / millimetres
G90 / G91	absolute / incremental

03 ZEROS AND COMPENSATION

CODE	FUNCTION
G28	return to reference point
G53	machine coordinates
G54-G59	work zero offset
G92	zero set in program
G40	cancel radius compensation
G41 / G42	compensation left / right
G43	tool length compensation

04 M-CODES

CODE	FUNCTION
M00 / M01	stop / optional stop
M03 / M04	spindle CW / CCW
M05	spindle stop
M06	tool change
M08 / M09	coolant on / off
M30	end of program, rewind

05 CYCLES AND FEED

CODE	FUNCTION
G33	threading, feed synchronised
G76	fine boring (mill) / thread (lathe)
G80	cancel cycle
G81	simple drilling
G82	drilling with dwell
G83	peck drilling
G84	tapping
G87	back boring
G94 / G95	feed mm/min / mm/rev
G96 / G97	constant vc / constant rpm
G98 / G99	return: initial / R plane

06 / BEFORE START

Before you run a program

- simulate the program and check for collisions
- G54 zero measured and entered
- G21 units and G90 mode at the start
- first part at reduced feed
- cycle numbers checked in the control manual

Cycle numbers and M-codes above M30 differ between controls (Fanuc, Siemens, Heidenhain). Always check the manual of the specific machine.