

Daniel's CNC programming Quick-Reference

(for Mach3 and EMC2) – Issue 1, 9 November 2011

To make it clear that the modal groups for G-codes are separate from the modal groups for M-codes, I've given the group numbers a prefix, and used lower-case so as not to confuse **group numbers** with **program codes** (though program codes are case-insensitive).

* **Mach3-only**, **EMC2-only** (where the meanings differ, I've quoted the *Mach3* meaning)

G Code*	Modal group	Description
G0	g1	Rapid travel; Follow by at least one of X, Y or Z value; e.g. G0 X50.0 Y90.0 Z5.0 ; In polar coordinates (see G16), X~ is the radius of the line from the G16 polar origin and Y~ is the angle in degrees (CCW from "3 o'clock").
G1	g1	Linear motion from current position; Follow by at least one of X, Y or Z value; e.g. G1 Z-1.0 . Polar as for G0.
G2	g1	Circular/arc interpolation (clockwise); e.g. G2 X40 Y70 I0 J-30 X, Y, Z = position of end-point, I, J, K = X/Y/Z offset from current position to centre; At least one of X, Y, Z and one of I, J, K required (dep. on plane)
G3	g1	Circular/arc interpolation (counter-clockwise); (Can also use R format for G2/3, but not recommended for arcs near 180° or 360°)
G4		Dwell; P = time in seconds or milliseconds; e.g. G4 P5 (seconds in <i>EMC2</i> ; in <i>Mach3</i> , set-up in Config for seconds or ms)
G10		Coordinate system origin setting (see manual)
G12		Clockwise circular pocket; I = offset from centre; e.g. G12 I3.0 The tool is moved in the X direction by the value of the I word (normally < tool diameter) and a circle cut in the direction specified, with the original X and Y coordinates as the centre. The tool is returned to the centre.
G13		Counter-clockwise circular pocket (see G12)
G15		Cancel polar coordinates (i.e. revert to Cartesian)
G16		Polar coordinate moves for G0/G1; The current coordinates are the temporary centre.
G17	g2	X-Y plane selection; Affects G2/3 and some canned cycles.
G18	g2	Z-X plane selection
G19	g2	Y-Z plane selection
G20	g6	Select inch units
G21	g6	Select millimetre units
G28		Return to 'home'
G28.1		Reference axes (see manual – don't use without home switches!)
G30		Return to 'home'

G Code*	Modal group	Description
G31		Straight probe touch function (see manual)
G40	g7	Cutter radius compensation cancel; It is OK to turn compensation off when it is already off.
G41	g7	Cutter radius compensation left ; e.g. G41 D2 The D word is optional; if there is no D word, the radius of the tool currently in the spindle will be used. If used, the D number should normally be the slot number of the tool in the spindle. (For 'left' or 'right', imagine sitting on top of the cutter and facing its direction of movement.)
G42	g7	Cutter radius compensation right ; e.g. G42 P3.175 G41 and G42 can be qualified by a P-word. This will override the value of the diameter of the tool (if any) given in the current tool table entry; NB: <i>Mach3</i> executes cutter compensation poorly for complex shapes.
G43	g8	Tool length compensation positive; e.g. G43 H3 H is the desired index in the tool table.
G49	g8	Tool length offset cancel
G50		Cancel scale factors (reset all axes to scale 1.0)
G51		Define a scale factor that will be applied to X, Y, Z, A, B, C, I & J words before they are used; e.g. G51 X1.1 Y1.1 Z1.1 A1.0 B1.0 C1.0 (not all need be spec'd), where the X/Y/Z etc words are the scale factors for the given axes.
G52		Offset the origin of the currently active coordinate system; X, Y, Z, A, B, C can be offset; at least one must be specified in this command; e.g. G52 X-7 moves the X-origin 7 units left (thus the X coordinate of the spindle would be 7 units greater than before; the spindle doesn't move). To reset the origin: G52 X0 (and similarly for other axes, if they were offset).
G53		Use machine coordinate system for current G0 or G1 move, but only for the current command line; e.g. G1 G53 X123.5 Y207.2
G54-59	g12	Select fixture offsets 1-6 respectively (see manual)
G61		Exact stop mode: the machine stops briefly at the end of each programmed move; the tool will therefore precisely follow the commanded path.
G64		Constant velocity mode: corners will be rounded (see manual)
G68		Rotate coordinate system; e.g. G68 A12 B25 I1 R45 where R = angle, A, B = X, Y centre of rotation; optional I (value is required , but ignored) adds R angle to any previous G68 rotation.
G69		Cancel coordinate system rotation
G70		Same as G20
G71		Same as G21
G73		Peck drill (see manual)
G80	g1	Cancel modal motion

G Code*	Modal group	Description
G81	g1	Drill, spot drill (see manual)
G82	g1	Drill, counterbore, with dwell (see manual); P = dwell time in seconds or milliseconds (see G4).
G83	g1	Deep hole cycle ('peck drilling') (see manual)
G84	g1	Right hand tap (see manual) – requires speed-feed synchronization!
G85	g1	Boring/reaming (see manual)
G86	g1	Boring/reaming with dwell & spindle stop (see manual)
G87	g1	Back boring (see manual)
G88	g1	Boring/reaming with dwell & spindle stop, manual retract (see manual)
G89	g1	Boring/reaming with dwell (see manual)
G90	g3	Absolute positioning (for X, Y, Z, A, B, C values; I and J numbers always represent increments)
G91	g3	Incremental positioning (for X, Y, Z, A, B, C values)
G92		Offset coordinates and set parameters (see manual)
G92.1		Cancel G92 and set parameters to zero (see manual)
G92.2		Cancel G92 but don't zero parameters (see manual)
G92.3		Set the axis offset values to the values given in parameters (see manual)
G93	g5	Specify F to be inverse time feed rate; move takes $1/F$ minutes (see manual)
G94	g5	Specify F to be units per minute feed rate (default)
G95	g5	Specify F to be units per revolution (of spindle) feed rate
G98	g10	Return to initial clearance plane after canned cycle
G99	g10	Return to R clearance plane after canned cycle; e.g. G99 R5.0

M Code*	Modal group	Description
M0	m4	Stop a running program temporarily (continues from next line, on restart)
M1	m4	Optional stop (only if the optional stop switch is on)
M2	m4	End of program (without rewind – see M30)
M3	m7	Spindle on clockwise
M4	m7	Spindle on counter-clockwise
M5	m7	Spindle off (Note: M2 and M30 will normally also stop the spindle)
M6	m6	Tool change; pauses program and calls macros (see manual)
M7	m8	Mist coolant on

M Code*	Modal group	Description
M8	m8	Flood coolant on
M9	m8	Coolant off
M26	m2	Enable automatic B-axis clamping
M27	m2	Disable automatic B-axis clamping
M30	m4	Program end and rewind
M47		Repeat program from first line
M48	m9	Enable speed and feed override
M49	m9	Disable speed and feed override
M60	m4	Pallet shuttle and program stop
M98		Execute subroutine (in same file or separate file); P = subroutine N°, L = N° of repetitions (defaults to 1 if omitted) e.g. M98 P4321 L18 or M98 (test.nc) L26
M99		End of subroutine ('return'), or re-start prog. execution from the first line

Other Codes

Code	Description
N	Line number – has no effect on execution, and not allowed on 'O' (name) lines
O	Program or subroutine 'name' (integers only, typically 1–4 digits); NB: 'Oh' not 'Zero'; e.g. O4321 (and see M98 example)
F	Set tool feed rate (usually units/min; see G93–95); e.g. F120
S	Set spindle speed (RPM); e.g. S4000
T	Select tool N°; e.g. T6

Variables and Expressions

In G-code, variables are called 'parameters', and are numbered from 1 to 5399 (*EMC2*) or 10320 (*Mach3*) (some parameter numbers have designated purposes – see **manual!**) They're used by prefixing a hash (#) to the parameter number, and are assigned a value in this way:

#11 = 23 (assigns the value 23 to parameter number 11)

Anywhere a value can be used, a parameter or an expression can be used instead. Expressions are contained inside square brackets, use the common mathematical and logical operators, and can be nested. Here's an example of the use of parameters and expressions:

```
#10 = 0 (starting angle)
M98 P1 L12 (drill 12 holes around a 50mm diam circle, centre 40,40)
O1 (subroutine)
#11 = [40 + [25 * cos[#10]]] (X for drilling)
#12 = [40 + [25 * sin[#10]]] (Y for drilling)
G0 X#11 Y#12 Z2 (go there)
... (some drilling operation)
#10 = [#10 + 30] (add 30 more degrees)
M99 (return)
```