# *Daniel's* CNC programming Quick-Reference

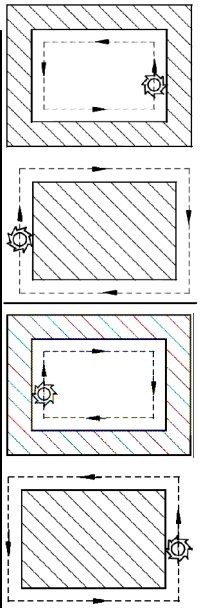
(for *Mach3* and *EMC2*) – Issue 6, 19 February 2014

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;<br>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;<br>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**<br>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**<br>(seconds in *EMC2*; in *Mach3*, 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**<br>The tool is moved in the X direction by the value of the I word 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. (See G13 for handy formula.) |
| G13 | | **Counter-clockwise** circular pocket (see G12). To bore a hole of diameter ***D*** using end/side-mill bit diameter ***d***, where $d < D \le 3d$, use $I = \frac{(D-d)}{2}$ |
| G15 | | Cancel polar coordinates (i.e. revert to Cartesian) |
| G16 | | Polar coordinate moves for G0/G1;<br>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' |
| G31 | | Straight probe touch function (see manual) |
| G40 | g7 | Cutter radius compensation cancel;<br>It is OK to turn compensation off when it is already off. |

| G–Code* | Modal group | Description |
|---|---|---|
| G41 | g7 | Cutter radius compensation **left**; e.g. **G41 D2** →<br>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** →<br>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;<br>NB: *Mach3* executes cutter compensation poorly for complex shapes. |
| G43 | g8 | Tool length compensation positive; e.g. **G43 H3**<br>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;<br>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**<br>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 with partial retract (see manual); |
| G80 | g1 | Cancel modal motion |
| G81 | g1 | Drill, spot drill (see manual) |
| G82 | g1 | Drill, counterbore, with dwell (see manual);<br>P = dwell time in seconds or milliseconds (see G4). |
| G83 | g1 | Deep hole cycle ('peck' drilling with full retract) (see manual);<br>e.g. **G83 X1.5 Y3.7 Z-1.6 R1.0 Q0.6** (R is Z retract level, Q is ΔZ) |
| G84 | g1 | Right hand tap (see manual) – **requires speed-feed synchronization!** |
| G85 | g1 | Boring/reaming (see manual) |



| G–Code* | Modal group | Description |
|---|---|---|
| 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 |
| 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 |

| M Code* | Modal group | Description |
|---|---|---|
| M98 | | Execute subroutine (in same file or separate file); ←<br>P = subroutine Nº, L = Nº of repetitions (defaults to 1 if omitted)<br>e.g. **M98 P4321 L18** or **M98 (test.nc) L26**<br>NB: In the second case, **test.nc** must be located in **…*****Mach3*****Subroutines** |
| 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);<br>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 Nº; e.g. **T6** |

# Unary operations

The unary operations are: **ABS** (absolute value), **ACOS** (arc cosine), **ASIN** (arc sine), **ATAN** (arc tangent), **COS** (cosine), **EXP** (*e* raised to the given power), **FIX** (round down), **FUP** (round up), **LN** (natural logarithm), **ROUND** (round to the nearest whole number), **SIN** (sine), **SQRT** (square root), and **TAN** (tangent). Arguments to unary operations that take angle measures (COS, SIN, and TAN) are in degrees. Values returned by unary operations that return angle measures (ACOS, ASIN, and ATAN) are also in degrees.

The FIX operation rounds towards the left (less positive or more negative) on a number line, so that **FIX[2.8]** = 2 and **FIX[-2.8]** = –3, for example. The FUP operation rounds towards the right (more positive or less negative) on a number line; **FUP[2.8]** = 3 and **FUP[-2.8]** = –2, for example.

# Variables and Expressions

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

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, unary and logical operators, and can be nested. Here's an example of the use of parameters and expressions:

```
#10 = 0                          (assign starting angle)
M98 P1 L12                       (drill 12 holes around a 50mm diam circle, centre 40,40)
M30                              (end and rewind)

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)
```