

Some the most common G CODE commands

G code is a system that describes vectorial motion and uses a single alphabet character followed by a number. Generally, there are two types of commands these are the G codes that mainly describe movements and the M command that describe miscellaneous commands. There are 100 M commands where M0 to M99 are pre-defined however 100 to 200 are available for custom commands and therefore are specific to a system or manufacturer.

G code can be written without spaces although this makes it difficult to read for a human, below are legal G code

```
G01x100y22.002z-1.9f22
```

The meaning of the above is dependent on the default setting of the machine or if there was a previous instruction that sets the measuring standard. In TRM controllers the measuring standard is set in the G code menu, however it is never a good idea to rely on defaults just in case machines have different default settings. Therefore, the above states a feed move at speed of 22 units per minute to X=100 Y=22.002 and Z=-1.9

The better way to write this is

```
F22
```

```
G01 X100 Y22.002 Z-1.9
```

M codes are normally used for things like controlling inputs or outputs such as M3 this sets the spindle in the clock wise direction, other commands turn on coolant etc. Other M codes such as M0 or M2 mark the end of the program. Below is a description of some common commands.

F (Feed rate parameter)

Creates a Feed Rate command that sets the speed of progression of the cutter in cutting commands such as G1. Note the feed rate is set in mm or inches per minute. For example F100 would set the speed to 100mm per minute if the machine is set in the metric mode or 100 inches per minute if the machine is set in the imperial mode. This command remains active until another F command is encountered.

G0 (Rapid Move)

Sets the system in the G0 Rapid move mode this is a persistent and remains until another command is enabled. Parameters include X, Y and Z to process the destinations for the X,Y and Z axis. The Rapid move uses the machines maximum speed and is used to move the tool between cutting areas (for example between pockets) or to the start of the cutting list or to the park position at the end of cutting. Note Rapid move must never be used for cutting! The maximum speed is set in the speed menu on TRM controllers.

Example g1 x100 y200 z50 // move to x100 y 200 and a z position of 50

G1 (Feed Move)

Sets the system in the G1 feed move mode this is a persistent and remains until another command is enabled. Parameters include X, Y and Z to process the destinations for the X,Y and Z axis.

Example g1 x100 y200 z50 // move to x100 y 200 and a z position of 50

G2 (Clockwise Arc)

Sets the system in the G2 clock wise arc mode this is a persistent and remains until another command is enabled.

G3 (Counter Clockwise Arc)

Sets the system in the G3 counter clockwise arc mode this is a persistent and remains until another command is enabled

General notes on G2 and G3:

Circular or helical arcs are specified by G2 (clockwise arc) or G3 (anticlockwise arc) at the current feed rate. The axis of the circle or helix must be parallel to the X, Y, or Z axis of the machine coordinate system. The by default XY is implied, this can be specified with G17 (XY-plane), G18 (XZ-plane), or G19 (YZ-plane). To program a helix, include the axis word perpendicular to the arc plane: for example, if in the G17 plane, include a Z word. This will cause the Z axis to move to the programmed value during the circular XY motion. To program more than one full turn, use the P word specifying the number of full turns plus the programmed arc. The P word must be an integer. If P is unspecified, the behaviour is as if P1 was given: that is, only one full or partial turn will result. For example, if a 180 degree arc is programmed with a P2, the resulting motion will be 1 1/2 rotations. For each P increment above 1 an extra full circle is added to the programmed arc.

The arc centre is absolute or relative as set by G90.1 or G91.1 respectively the default being relative. There are two formats are allowed for specifying an arc: these are the normal Centre Format and Radius Format.

G2/G3 Centre Format Arcs

Centre format arcs are more common and are the preferred format to use for most CNC systems. The end point of the arc along with the offset to the centre of the arc from the current location are used to program arcs that are less than a full circle. It is OK if the end point of the arc is the same as the current location as in a 360 or more rotation.

Parameters used with centre arc format

XY plane

- X Specifies the End point in the X plane.
- Y Specifies the End point in the Y plane.
- I Specifies Y centre point offset.
- J Specifies X centre point offset.
- Z Helix .
- P Number of turns.

XZ Plane

- X Specifies the End point in the X plane.
- Z Specifies the End point in the Z plane.
- Y Helix.
- K Specifies Z centre point offset.
- J Specifies X centre point offset.
- Y Helix .
- P Number of turns.

YZ Plane

- Y Specifies the End point in the Y plane.
- Z Specifies the End point in the Z plane.
- K Specifies Z centre point offset.

- I Specifies Y centre point offset.
- Y Helix .
- **P Number of turns.**

The offset to the centre of the arc from the current location and optionally the number of turns are used to program full circles. Remember errors can occur due to rounding therefore 3 digit accuracy is advised for metric systems and 4 four imperial.

Incremental Arc Distance Mode (relative):

Arc centre offsets are a relative distance from the start location of the arc. Incremental Arc Distance Mode is default. One or more axis words and one or more offsets must be programmed for an arc that is less than 360 degrees. No axis words and one or more offsets must be programmed for full circles. The P word defaults to 1 and is optional. Use G91.1 to set the Incremental Arc Distance Mode.

Absolute Arc Distance Mode:

Arc centre offsets are the absolute distance from the current 0 position of the axis (the origin). One or more axis words and both offsets must be programmed for arcs less than 360 degrees. No axis words and both offsets must be programmed for full circles. The P word defaults to 1 and is optional.

The 'Absolute Arc Distance Mode is set by the G90.1 command.

G2/G3 Radius Format Arcs

This mode should not be used to specify arcs that are or approximate circles or semicircles. Other size arcs (in the range tiny to 165 degrees or 195 to 345 degrees) are OK. In the radius format, the coordinates of the end point of the arc in the selected plane are specified along with the radius of the arc. Program G2 axes R- (or use G3 instead of G2). R is the radius. The axis words are all optional except that at least one of the two words for the axes in the selected plane must be used. The R number is the radius as a floating point. A positive radius indicates that the arc turns through less than 180 degrees, while a negative radius indicates a turn of more than 180 degrees. If the arc is helical, the value of the end point of the arc on the coordinate axis parallel to the axis of the helix is also specified.

G2 X10 Y15 R20 Z5 (*radius format with arc*)

G4 (Dwell)

G4 is a dwell command implemented immediately and non-persistent G code command such as G4 dwell have a delay in seconds after the command such as G4 P0.5 for 0.5 seconds this function searches for the P and then the delay value on success waits and returns the number of chars to advance the buffer pointer if the P is not found or the delay is negative returns is a failure.

G7 (diameter mode)

Program G7 to enter the diameter mode for axis X on a lathe. When in the diameter mode the X axis moves on a lathe will be 1/2 the distance to the centre of the lathe. For example, X1 would move the cutter to 0.500" from the centre of the lathe thus giving a 1" diameter part.

G8 (Radius Mode)

Program G8 to enter the radius mode for axis X on a lathe. When in Radius mode the X axis moves on a lathe will be the distance from the centre. Thus, a cut at X1 would result in a part that is 2 in diameter. G8 is default at power up.

G17 (XY Planes)

Sets the system to XY arc planes.

G18 (XZ Planes)

Sets the system to XZ arc planes.

G19 (YZ Planes)

Sets the system to YZ arc planes.

G20 (use imperial)

Program G20 use inches for measurement.

G21 (uses Metric)

Program G20 use MM for measurement.

G28 (park)

Function G28 causes the system to park at the location set in the G Code menu then calls the map PARK command to park the axis and stop the machine to allow operator access to the work piece

G40 (Radius compensation off)

Program G40 turns off the radius compensation (radius compensation is not active currently)

G49 (tool length compensation cancel)

Tool length offset compensation cancel.

G50 (scaling off)

Scaling function cancel.

G53(absolute Mode on)

Sets the controller is an absolute mode where the datums are ignored for example if the X datum is 150 and X50 will cause a X position of 150 + 50 After this command X50 will result in a position of 50 because the datums will be ignored for 1 move This command is non-Modal and must be called each time an absolute move is required.

G61(Exact Path mode)

Exact path mode. G61 visits the programmed point exactly, even though that means temporarily coming to a complete stop

G64 (exact path mode)

Exact path mode. G61 visits the programmed point exactly, even though that means temporarily coming to a complete stop.

G64 (performance)

P - motion blending tolerance

Q - naive cam tolerance

G64 - best possible speed.

G64 P- <Q- > blending with tolerance.

G64 - without P means to keep the best speed possible, no matter how far away from

the programmed point you end up.

G64 P- Q- - is a way to fine tune your system for best compromise between speed and accuracy. The P- tolerance means that the actual path will be no more than P- away from the programmed endpoint. The velocity will be reduced if needed to maintain the path. In addition, when you activate G64 P- Q- it turns on the naive cam detector; when there are a series of linear XYZ feed moves at the same feed rate that are less than Q- away from being collinear, they are collapsed into a single linear move. On G2/G3 moves in the G17 (XY) plane when the maximum deviation of an arc from a straight line is less than the G64 P- tolerance the arc is broken into two lines (from start of arc to midpoint, and from midpoint to end). those lines are then subject to the naive cam algorithm for lines. Thus, line-arc, arc-arc, and arc-line cases as well as line-line benefit from the naive cam detector. This improves contouring performance by simplifying the path. It is OK to program for the mode that is already active. See also the Trajectory Control Section for more information on these modes. If Q is not specified then it will have the same behaviour as before and use the value of P-.

G80 (cancel canned cycles)

Cancels canned cycles

G81 (cycle drilling command)

G81 cycle drilling command Data X,Y,Z,R,L where X = the Absolute x position or the X increment in the relative mode Y = the Absolute Y position or the Y increment in the relative mode Z = the Absolute Z position R = the Absolute R position this is the point at which the Z axis changes from rapid move to feed move L is the number of loops or repetitions works only in the relative mode

In the absolute mode works as follows: 1) does a rapid move to position X,Y 2) does a rapid move to position R 3) does a feed move to Z

In the relative mode works as follows: 1) does a rapid move to relative position X,Y (e.g. current and X and Y position + X and Y respectively) 2) does a rapid move to position R 3) does a feed move to Z 4) if L is greater than 1 decrements L and repeats lines 1 to 4

G82 (cycle drilling command)

G82 cycle drilling command Data X,Y,Z,R,L where X = the Absolute x position or the X increment in the relative mode Y = the Absolute Y position or the Y increment in the relative mode Z = the Absolute Z position R = the Absolute R position this is the point at which the Z axis changes from rapid move to feed move L = the number of loops or repetitions works only in the relative mode P = the dwell time after the drill has reached final position in seconds.

In the absolute mode works as follows: 1) does a rapid move to position X,Y 2) does a rapid move to position R 3) does a feed move to Z

In the relative mode works as follows: 1) does a rapid move to relative position X,Y (e.g. current and X and Y position + X and Y respectively) 2) does a rapid move to position R 3) does a feed move to Z 4) if L is greater than 1 decrements L and repeats lines 1 to 4

G90 (absolute measurement mode)

Absolute distance mode In absolute distance mode, axis numbers (X, Y, Z, A, B, C, U, V, W) usually represent positions in terms of the currently active coordinate system. Any exceptions

to that rule are described explicitly

G90_1 (absolute distance mode)

Absolute distance mode for I, J & K offsets. When G90.1 is in effect I and J both must be specified with G2/3 for the XY plane or J and K for the XZ plane or it is an err.

G91 (incremental distance mode)

Incremental distance mode In incremental distance mode, axis numbers usually represent increments from the current coordinate buffer

G91_1

Incremental distance mode for I, J & K offsets. G91.1 Returns I, J & K to their default behaviour.

G94 (set the feed rate to units per minute)

Set the feed rate to units per minute implemented as a stub (not active included to cause an error during g code loading.

G96 (Constant surface speed)

Sets the constant surface speed mode. G96 D- S- - selects constant surface speed of S feet per minute (if G20 is in effect) or millimetres per minute (if G21 is in effect). D- is optional. When using G96, ensure that X0 in the current coordinate system (including offsets and tool lengths) is the center of rotation or will not give the desired spindle speed. G96 is not affected by radius or diameter mode.

G97 (spindle operational mode)

Spindle RPM mode.

I (x offset)

I DIMENSION USED AS X OFFSET IN ARCS AND IN CANNED CYCLES I immediate command.

J (Y offset)

J DIMENSION USED AS Y OFFSET IN ARCS AND IN CANNED CYCLES this command is an immediate.

M0 (program pause)

M0 Program Pause, On TRM controllers causes a map press_run command to be written, this requires a button to be pressed on the keypad.

M2 (end of program)

End Program M2 or M30.

M3 (spindle clock wise start)

M3 turn spindle clock wise.

M4 (spindle counter clock wise start)

M4 Turn spindle counter clock wise.

M5 (stop spindle)

M5 Stop spindle.

M6 (change tool)

Creates a tool command in map and passes the number to the tool changer or invokes a map tool program.

M7 (Mist coolant on)

M8 (Turn flood coolant on)

M9 (Turn all coolant off)

M60 (PalLet Change Pause)

N (line numbers)

Line (block) numbers: Optional, so often omitted. Necessary for certain tasks, such as M99 P address (to tell the control which block of the program to return to if not the default one) or GoTo statements (if the control supports those). N numbering need not increment by 1 (for example, it can increment by 10, 20, or 1000) and can be used on every block or only in certain spots throughout a program. System parameter number: G10 allows changing of system parameters under program control.

P (dwell or number of turns parameter)

P reads the P parameter used for the number of turns in the G2/3 command or G4 dwell.

R (normally used for radius)

R DIMENSION Radius arc immediate command.

S (spindle speed)

Reads the S parameter used spindle speed mainly in the G97.

T (Tool number)

T reads the T parameter used in tool or analogue commands.

X (X position parameter)

X DIMENSION this command is an immediate command used to set the var x.

Y (Y position parameter)

Y DIMENSION this command is an immediate command used to set the var y.

Z (Z position parameter)

Z DIMENSION this command is an immediate command used to set the var Z.

The above is provided as a general guide to some of the most commonly used G code commands. Some controllers may not respond to all the commands above and the performance of each command is not guaranteed. This guide is provided without guarantee or liability by TRM Electronics Ltd.