G0 Rapid positioning

G1 Linear interpolation

G2 Clockwise circular/helical interpolation

G3 Counterclockwise circular/Helical interpolation

G4 Dwell

G10 Coordinate system origin setting

G12 Clockwise circular pocket

G13 Counterclockwise circular pocket

G15/G16 Polar Coordinate moves in G0 and G1

G17 XY Plane select

G18 XZ plane select

G19 YZ plane select

G20/G21 Inch/Millimeter unit

G28 Return home

G28.1 Reference axes

G30 Return home

G31 Straight probe

G40 Cancel cutter radius compensation

G41/G42 Start cutter radius compensation left/right

G43 Apply tool length offset (plus)

G49 Cancel tool length offset

G50 Reset all scale factors to 1.0

G51 Set axis data input scale factors

G52 Temporary coordinate system offsets

G53 Move in absolute machine coordinate system

G54 Use fixture offset 1

G55 Use fixture offset 2

G56 Use fixture offset 3

G57 Use fixture offset 4

G59 Use fixture offset 6 / use general fixture number

G61/G64 Exact stop/Constant Velocity mode

G68/G69 Rotate program coordinate system

G70/G71 Inch/Millimeter unit

G73 Canned cycle - peck drilling

G80 Cancel motion mode

G81 Canned cycle - drilling

G82 Canned cycle - drilling with dwell

G83 Canned cycle - peck drilling

G85/G86/G88/G89 Canned cycle - boring

G90 Absolute distance mode

G90.1 Absolute IJK mode

G91 Incremental distance mode

G91.1 Incremental IJK mode

G92 Offset coordinates and set parameters

G92.x Cancel G92 etc.

G93 Inverse time feed mode

G94 Units Per Min.

G98 Rapid Height By Z Height

G99 Rapid Height By R Height

G00 Rapid Move

(a) For rapid linear motion, program G0 X~ Y~ Z~ A~ B~ C~, where all the axis words are optional, except that at least one must be used. The G0 is optional if the current motion mode is G0. This will produce co-ordinated linear motion to the destination point at the current traverse rate (or slower if the machine will not go that fast). It is expected that cutting will not take place when a G0 command is executing.

(b) If G16 has been executed to set a Polar Origin then for rapid linear motion to a point described by a radius and angle G0 X~ Y~ can be used. X~ is the radius of the line from the G16 polar origin and Y~ is the angle in degrees measured with increasing values counterclockwise from the 3 o&p12/G-codes/pages.html#39;clock direction (i.e. the conventional four quadrant conventions).

Coordinates of the current point at the time of executing the G16 are the polar origin.

It is an error if:

¨ all axis words are omitted.

If cutter radius compensation is active, the motion will differ from the above; see Cutter Compensation. If G53 is programmed on the same line, the motion will also differ; see Absolute Coordinates.

Return to G-code list

G01 Linear Move

(a) For linear motion at feed rate (for cutting or not), program G1 X~ Y~ Z~ A~ B~ C~, where all the axis words are optional, except that at least one must be used. The G1 is optional if the current motion mode is G1. This will produce co-ordinated linear motion to the destination point at the current feed rate (or slower if the machine will not go that fast).

(b) If G16 has been executed to set a polar origin then linear motion at feed rate to a point described by a radius and angle G0 X~ Y~ can be used. X~ is the radius of the line from the G16 polar origin and Y~ is the angle in degrees measured with increasing values counterclockwise from the 3 o'clock direction (i.e. the conventional four quadrant conventions).

Coordinates of the current point at the time of executing the G16 are the polar origin.

It is an error if:

¨ all axis words are omitted.

If cutter radius compensation is active, the motion will differ from the above; see Cutter Compensation. If G53 is programmed on the same line, the motion will also differ; see Absolute Coordinates.

Return to G-code list

G02 & G03 Arc Move

A circular or helical arc is specified using either G2 (clockwise arc) or G3 (counterclockwise arc). The axis of the circle or helix must be parallel to the X, Y, or Z-axis of the machine coordinate system. The axis (or, equivalently, the plane perpendicular to the axis) is selected with G17 (Z-axis, XY-plane), G18 (Y-axis, XZ-plane), or G19 (X-axis, YZ-plane). If the arc is circular, it lies in a plane parallel to the selected plane.

If a line of code makes an arc and includes rotational axis motion, the rotational axes turn at a constant rate so that the rotational motion starts and finishes when the XYZ motion starts and finishes. Lines of this sort are hardly ever programmed.

If cutter radius compensation is active, the motion will differ from the above; see Cutter Compensation.

Two formats are allowed for specifying an arc. We will call these the center format and the radius format. In both formats the G2 or G3 is optional if it is the current motion mode.

Arc Center Format

In the center format, the coordinates of the end point of the arc in the selected plane are specified along with the offsets of the center of the arc from the current location. In this format, it is OK if the end point of the arc is the same as the current point. It is an error if:

¨ when the arc is projected on the selected plane, the distance from the current point to the center differs from the distance from the end point to the center by more than 0.0002 inch (if inches are being used) or 0.002 millimetre (if millimetres are being used).

The center is specified using the I and J words. There are two ways of interpreting them. The usual way is that I and J are the center relative to the current point at the start of the arc. This is sometimes called Incremental IJ mode. The second way is that I and J specify the center as actual coordinates in the current system. This is rather misleadingly called Absolute IJ mode. The IJ mode is set using the Configure>State... menu when Mach3 is set up. The choice of modes are to provide compatibility with commercial controllers. You will probably find Incremental to be best. In Absolute it will, of course usually be necessary to use both I and J words unless by chance the arc's centre is at the origin.

When the XY-plane is selected, program G2 X~ Y~ Z~ A~ B~ C~ I~ J~ (or use G3 instead of G2). The axis words are all optional except that at least one of X and Y must be used. I and J are the offsets from the current location or coordinates - depending on IJ mode (X and Y directions, respectively) of the center of the circle. I and J are optional except that at least one of the two must be used. It is an error if:

- **X and Y are both omitted,**
- **I and J are both omitted.**

When the XZ-plane is selected, program G2 X~ Y~ Z~ A~ B~ C~ I~ K~ (or use G3 instead of G2). The axis words are all optional except that at least one of X and Z must be used. I and K are the offsets from the current location or coordinates - depending on IJ mode (X and Z directions, respectively) of the center of the circle. I and K are optional except that at least one of the two must be used. It is an error if:

- **X and Z are both omitted,**
- **I and K are both omitted.**

When the YZ-plane is selected, program G2 X~ Y~ Z~ A~ B~ C~ J~ K~ (or use G3 instead of G2). The axis words are all optional except that at least one of Y and Z must be used. J and K are the offsets from the current location or coordinates - depending on IJ mode (Y and Z directions, respectively) of the center of the circle. J and K are optional except that at least one of the two must be used. It is an error if:

- **Y and Z are both omitted,**
- **J and K are both omitted.**

Here is an example of a center format command to mill an arc in Incremental IJ mode:

G17 G2 x10 y16 i3 j4 z9

That means to make a clockwise (as viewed from the positive z-axis) circular or helical arc whose axis is parallel to the Z-axis, ending where X=10, Y=16, and Z=9, with its center offset in the X direction by 3 units from the current X location and offset in the Y direction by 4 units from the current Y location. If the current location has X=7, Y=7 at the outset, the center will be at X=10, Y=11. If the starting value of Z is 9, this is a circular arc; otherwise it is a helical arc. The radius of this arc would be 5.

The above arc in Absolute IJ mode would be:

G17 G2 x10 y16 i10 j11 z9

In the center format, the radius of the arc is not specified, but it may be found easily as the distance from the center of the circle to either the current point or the end point of the arc.

Return to G-code list

G4 Dwell

For a dwell, program G4 P~ . This will keep the axes unmoving for the period of time in seconds or milliseconds specified by the P number. The time unit to be used is set up on the Config>Logic dialog. For example, with units set to Seconds, G4 P0.5 will dwell for half a second. It is an error if:

- **the P number is negative.**

Return to G-code list

G10 Tool Offset and Work Offset Tables

To set the offset values of a tool, program
G10 L1 P~ X~ Z~ A~, where the P number must evaluate to an integer in the range 0 to 255 - the tool number - Offsets of the tool specified by the P number are reset to the given. The A

number will reset the tool tip radius. Only those values for which an axis word is included on the line will be reset. The Tool diameter cannot be set in this way.

To set the coordinate values for the origin of a fixture coordinate system, program G10 L2 P~ X~ Y~ Z~ A~ B~ C~, where the P number must evaluate to an integer in the range 1 to 255 - the fixture number - (Values 1 to 6 corresponding to G54 to G59) and all axis words are optional. The coordinates of the origin of the coordinate system specified by the P number are reset to the coordinate values given (in terms of the absolute coordinate system). Only those coordinates for which an axis word is included on the line will be reset.

It is an error if:

- the P number does not evaluate to an integer in the range 0 to 255.

If origin offsets (made by G92 or G92.3) were in effect before G10 is used, they will continue to be in effect afterwards.

The coordinate system whose origin is set by a G10 command may be active or inactive at the time the G10 is executed.

The values set will not be persistent unless the tool or fixture tables are saved using the buttons on Tables screen.

Example: G10 L2 P1 x3.5 y17.2 sets the origin of the first coordinate system (the one selected by G54) to a point where X is 3.5 and Y is 17.2 (in absolute coordinates). The Z coordinate of the origin (and the coordinates for any rotational axes) are whatever those coordinates of the origin were before the line was executed.

Return to G-code list

G12 & G13 CW/CCW Circular Pocket

These circular pocket commands are a sort of canned cycle which can be used to produce a circular hole larger than the tool in use or with a suitable tool (like a woodruff key cutter) to cut internal grooves for "O" rings etc.

Program G12 I~ for a clockwise move and G13 I~ for a counterclockwise move.

The tool is moved in the X direction by the value if 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.

Its effect is undefined if the current plane is not XY.

Return to G-code list

G15 & G16 Exit and Enter Polar Mode

It is possible for G0 and G1 moves in the X/Y plane only to specify coordinates as a radius and angle relative to a temporary center point. Program G16 to enter this mode. The current coordinates of the controlled point are the temporary center.

Program G15 to revert to normal Cartesian coordinates.

G0 X10 Y10 // normal G0 move to 10,10
G16 //start of polar mode.
G10X10Y45
(this will move to X 17.xxx, Y 17.xxx which is a spot on a circle) (of radius 10 at 45 degrees from the initial coordinates of 10,10.)

This can be very useful, for example, for drilling a circle of holes. The code below moves to a circle of holes every 10 degrees on a circle of radius 50 mm centre X = 10, Y = 5.5 and peck drills to Z = -0.6

G21 // metric
G0 X10Y5.5
G16
G1 X50 Y0 //polar move to a radius of 50 angle 0deg
G83 Z-0.6 // peck drill
G1 Y10 // ten degrees from original center...
G83 Z-0.6
G1 Y20 // 20 degrees....etc...

G1 Y30

G1 Y40

> ...etc....

G15 //back to normal cartesian

Notes:

(1) you must not make X or Y moves other than by using G0 or G1 when G16 is active

(2) This G16 is different to a Fanuc implementation in that it uses the current point as the polar center. The Fanuc version requires a lot of origin shifting to get the desired result for any circle not centred on 0,0

Return to G-code list

G17,G18 & G19 Plane Selection

Program G17 to select the XY-plane, G18 to select the XZ-plane, or G19 to select the YZ-plane. The effects of having a plane selected are discussed in under G2/3 and Canned cycles

Return to G-code list

G20 & G21 Unit Selection

Program G20 to use inches for length units. Program G21 to use millimetres.

It is usually a good idea to program either G20 or G21 near the beginning of a program before any motion occurs, and not to use either one anywhere else in the program. It is the responsibility of the user to be sure all numbers are appropriate for use with the current length units. See also G70/G71 which are synonymous.

Return to G-code list

G28 & G30 Return to Home

A home position is defined (by parameters 5161-5166). The parameter values are in terms of the absolute coordinate system, but are in unspecified length units.

To return to home position by way of the programmed position, program G28 X~ Y~ Z~ A~ B~ C~ (or use G30). All axis words are optional. The path is made by a traverse move from the current position to the programmed position, followed by a traverse move to the home position. If no axis words are programmed, the intermediate point is the current point, so only one move is made.

Return to G-code list

G28.1 Reference Axis

Program G28.1 X~ Y~ Z~ A~ B~ C~ to reference the given axes. The axes will move at the current feed rate towards the home switch(es), as defined by the Configuration. When the absolute machine coordinate reaches the value given by an axis word then the feed rate is set to that defined by Configure>Config Referencing. Provided the current absolute position is approximately correct, then this will give a soft stop onto the reference switch(es).

Return to G-code list

G31 Straight Probe

Program G31 X~ Y~ Z~ A~ B~ C~ to perform a straight probe operation. The rotational axis words are allowed, but it is better to omit them. If rotational axis words are used, the numbers must be the same as the current position numbers so that the rotational axes do not move. The linear axis words are optional, except that at least one of them must be used. The tool in the spindle must be a probe.

It is an error if:

- **the current point is less than 0.254 millimetre or 0.01 inch from the programmed point.**
- **G31 is used in inverse time feed rate mode,**
- **any rotational axis is commanded to move,**

¨ no X, Y, or Z-axis word is used.

In response to this command, the machine moves the controlled point (which should be at the end of the probe tip) in a straight line at the current feed rate toward the programmed point. If the probe trips, the probe is retracted slightly from the trip point at the end of command execution. If the probe does not trip even after overshooting the programmed point slightly, an error is signalled.

After successful probing, parameters 2000 to 2005 will be set to the coordinates of the location of the controlled point at the time the probe tripped and a triplet giving X, Y and Z at the trip will be written to the triplet file if it has been opened by the M40 macro/OpenDigFile() function (q.v.)

Straight Probe Command

Using the straight probe command, if the probe shank is kept nominally parallel to the Z-axis (i.e., any rotational axes are at zero) and the tool length offset for the probe is used, so that the controlled point is at the end of the tip of the probe:

without additional knowledge about the probe, the parallelism of a face of a part to the XY-plane may, for example, be found.

if the probe tip radius is known approximately, the parallelism of a face of a part to the YZ or XZ-plane may, for example, be found.

if the shank of the probe is known to be well-aligned with the Z-axis and the probe tip radius is known approximately, the center of a circular hole, may, for example, be found.

if the shank of the probe is known to be well-aligned with the Z-axis and the probe tip radius is known precisely, more uses may be made of the straight probe command, such as finding the diameter of a circular hole.

If the straightness of the probe shank cannot be adjusted to high accuracy, it is desirable to know the effective radii of the probe tip in at least the +X, -X, +Y, and -Y directions. These quantities can be stored in parameters either by being included in the parameter file or by being set in a Mach3 program.

Using the probe with rotational axes not set to zero is also feasible. Doing so is more complex than when rotational axes are at zero, and we do not deal with it here.

Example Code

As a usable example, the code for finding the center and diameter of a circular hole is shown in figure 11.5. For this code to yield accurate results, the probe shank must be well-aligned with the Z-axis, the cross section of the probe tip at its widest point must be very circular, and the probe tip radius (i.e., the radius of the circular cross section) must be known precisely. If the probe tip radius is known only approximately (but the other conditions hold), the location of the hole center will still be accurate, but the hole diameter will not.

```
N010 (probe to find center and diameter of circular hole)

N020 (This program will not run as given here. You have to)

N030 (insert numbers in place of .)

N040 (Delete lines N020, N030, and N040 when you do that.)

N050 G0 Z F

N060 #1001=

N070 #1002=

N080 #1003=

N090 #1004=

N100 #1005=[/2.0 - #1004]

N110 G0 X#1001 Y#1002 (move above nominal hole center)

N120 G0 Z#1003 (move into hole - to be cautious, substitute G1 for G0 here)

N130 G31 X[#1001 + #1005] (probe +X side of hole)

N140 #1011=#2000 (save results)

N150 G0 X#1001 Y#1002 (back to center of hole)

N160 G31 X[#1001 - #1005] (probe -X side of hole)
```

```
N170 #1021=[[#1011 + #2000] / 2.0] (find pretty good X-value of hole center)

N180 G0 X#1021 Y#1002 (back to center of hole)

N190 G31 Y[#1002 + #1005] (probe +Y side of hole)

N200 #1012=#2001 (save results)

N210 G0 X#1021 Y#1002 (back to center of hole)

N220 G31 Y[#1002 - #1005] (probe -Y side of hole)

N230 #1022=[[#1012 + #2001] / 2.0] (find very good Y-value of hole center)

N240 #1014=[#1012 - #2001 + [2 * #1004]] (find hole diameter in Y-direction)

N250 G0 X#1021 Y#1022 (back to center of hole)

N260 G31 X[#1021 + #1005] (probe +X side of hole)

N270 #1031=#2000 (save results)

N280 G0 X#1021 Y#1022 (back to center of hole)

N290 G31 X[#1021 - #1005] (probe -X side of hole)

N300 #1041=[[#1031 + #2000] / 2.0] (find very good X-value of hole center)

N310 #1024=[#1031 - #2000 + [2 * #1004]] (find hole diameter in X-direction)

N320 #1034=[[#1014 + #1024] / 2.0] (find average hole diameter)

N330 #1035=[#1024 - #1014] (find difference in hole diameters)

N340 G0 X#1041 Y#1022 (back to center of hole)

N350 M2 (that's all, folks)
```

Figure 10.5 - Code to Probe Hole

In figure 10.5 an entry of the form is meant to be replaced by an actual number that matches the description of number. After this section of code has executed, the X-value of

the center will be in parameter 1041, the Y-value of the center in parameter 1022, and the diameter in parameter 1034. In addition, the diameter parallel to the X-axis will be in parameter 1024, the diameter parallel to the Y-axis in parameter 1014, and the difference (an indicator of circularity) in parameter 1035. The probe tip will be in the hole at the XY center of the hole.

The example does not include a tool change to put a probe in the spindle. Add the tool change code at the beginning, if needed.

Return to G-code list

G40,G41 & G42 Cutter Comp

To turn cutter radius compensation off, program G40. It is OK to turn compensation off when it is already off.

Cutter radius compensation may be performed only if the XY-plane is active.

To turn cutter radius compensation on left (i.e., the cutter stays to the left of the programmed path when the tool radius is positive), program G41 D~ To turn cutter radius compensation on right (i.e., the cutter stays to the right of the programmed path when the tool radius is positive), program G42 D~ 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, although this is not required. It is OK for the D number to be zero; a radius value of zero will be used.

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.

It is an error if:

- **the D number is not an integer, is negative or is larger than the number of carousel slots,**
- **the XY-plane is not active,**
- **cutter radius compensation is commanded to turn on when it is already on.**

The behavior of the machining system when cutter radius compensation is ON is described in the chapter of Cutter Compensation. Notice the importance of programming valid entry and exit moves.

Return to G-code list

G43,G44 & G49 Tool Length Offsets

To use a tool length offset, program G43 H~, where the H number is the desired index in the tool table. It is expected that all entries in this table will be positive. The H number should be, but does not have to be, the same as the slot number of the tool currently in the spindle. It is OK for the H number to be zero; an offset value of zero will be used. Omitting H has the same effect as a zero value.

G44 is provided for compatibility and is used if entries in the table give negative offsets.

It is an error if:

- **the H number is not an integer, is negative, or is larger than the number of carousel slots.**

To use no tool length offset, program G49

It is OK to program using the same offset already in use. It is also OK to program using no tool length offset if none is currently being used.

Return to G-code list

G50 & G51 Scale Factor

To define a scale factor which will be applied to an X, Y, Z, A, B, C, I & J word before it is used program G51 X~ Y~ Z~ A~ B~ C~ where the X, Y, Z etc. words are the scale factors for the given axes. These values are, of course, never themselves scaled.

It is not permitted to use unequal scale factors to produce elliptical arcs with G2 or G3.

To reset the scale factors of all axes to 1.0 program G50

Return to G-code list

G52 Coordinate System Offset

To offset the current point by a given positive or negative distance (without motion), program
G52 X~ Y~ Z~ A~ B~ C~ , where the axis words contain the offsets you want to provide. All axis words are optional, except that at least one must be used. If an axis word is not used for a given axis, the coordinate on that axis of the current point is not changed. It is an error if:

- **all axis words are omitted.**

G52 and G92 use common internal mechanisms in Mach3 and may not be used together.

When G52 is executed, the origin of the currently active coordinate system moves by the values given.

The effect of G52 is cancelled by programming G52 X0 Y0 etc.

Here is an example. Suppose the current point is at X=4 in the currently specified coordinate system, then G52 X7 sets the X-axis offset to 7, and so causes the X-coordinate of the current point to be -3.

The axis offsets are always used when motion is specified in absolute distance mode using any of the fixture coordinate systems. Thus all fixture coordinate systems are affected by G52.

Return to G-code list

G53 Move in ABS Coordinates

Absolute machine coordinates:G53 - move in" For linear motion to a point expressed in absolute coordinates, program G1 G53 X~ Y~ Z~ A~ B~ C~ (or similarly with G0 instead of G1), where all the axis words are optional, except that at least one must be used. The G0 or G1 is optional if it is in the current motion mode. G53 is not modal and must be programmed

on each line on which it is intended to be active. This will produce co-ordinated linear motion to the programmed point. If G1 is active, the speed of motion is the current feed rate (or slower if the machine will not go that fast). If G0 is active, the speed of motion is the current traverse rate (or slower if the machine will not go that fast).

It is an error if:

¨ G53 is used without G0 or G1 being active,

¨ G53 is used while cutter radius compensation is on.

See relevant chapter for an overview of coordinate systems.

Return to G-code list

G54-G59 and G59 P1-254 Work Offsets

To select work offset #1, program G54, and similarly for the first six offsets. The system-number-G-code pairs are: (1-G54), (2-G55), (3-G56), (4-G57), (5-G58), (6-G59)

To access any of the 254 work offsets (1 - 254) program G59 P~ where the P word gives the required offset number. Thus G59 P5 is identical in effect to G58.

It is an error if:

¨ one of these G-codes is used while cutter radius compensation is on.

See relevant chapter for an overview of coordinate systems.

Return to G-code list

G61 & G64 Path Control Mode

Program G61 to put the machining system into exact stop mode, or G64 for constant velocity mode. It is OK to program for the mode that is already active. These modes are described in detail above.

Return to G-code list

G68 & G69 Rotate Coordinate System

Program G68 A~ B~ I~ R~ to rotate the program coordinate system.

A~ is the X coordinate and B~ the Y coordinate of the center of rotation in the current coordinate system (i.e. including all work and tool offsets and G52/G92 offsets.)

R~ is the rotation angle in degrees (positive is CCW viewed from the positive Z direction).

I~ is optional and the value is not used. If I~ is present it causes the given R value to be added to any existing rotation set by G68.

e.g. G68 A12 B25 R45 causes the coordinate system to be rotated by 45 degrees about the point Z=12, Y=25

Subsequently: G68 A12 B35 I1 R40 leaves the coordinate system rotated by 85 degrees about X = 12, Y=25

Program G69 to cancel rotation.

Notes:

- **This code only allows rotation when the current plane is X-Y**
- **The I word can be used even if the center point is different from that used before although, in this case, the results need careful planning. It could be useful when simulating engine turning.**

Return to G-code list

G70 & G71 Units

Program G70 to use inches for length units. Program G71 to use millimetres.

It is usually a good idea to program either G70 or G71 near the beginning of a program before any motion occurs, and not to use either one anywhere else in the program. It is the responsibility of the user to be sure all numbers are appropriate for use with the current length units. See also G20/G21 which are synonymous and preferred.

Return to G-code list

G73 High Speed Peck Drill

The G73 cycle is intended for deep drilling or milling with chip breaking. See also G83. The retracts in this cycle break the chip but do not totally retract the drill from the hole. It is suitable for tools with long flutes which will clear the broken chips from the hole. This cycle takes a Q number which represents a "delta" increment along the Z-axis. Program

G73 X~ Y~ Z~ A~ B~ C~ R~ L~ Q~

- **Preliminary motion, as described in G81 to 89 canned cycles.**
- **Move the Z-axis only at the current feed rate downward by delta or to the Z position, whichever is less deep.**
- **Rapid back out by the distance defined in the G73 Pullback DRO on the Settings screen.**
- **Rapid back down to the current hole bottom, backed off a bit.**
- **Repeat steps 1, 2, and 3 until the Z position is reached at step 1.**
- **Retract the Z-axis at traverse rate to clear Z.**

It is an error if:

¨ **the Q number is negative or zero.**

Return to G-code list

G80 Cancel Canned Cycles

Program G80 to ensure no axis motion will occur. It is an error if:

- **Axis words are programmed when G80 is active, unless a modal group 0 G code is programmed which uses axis words.**

Return to G-code list

Return to G-code list

G81 - G89 Cannend Cycles

The canned cycles G81 through G89 have been implemented as described in this section. Two examples are given with the description of G81 below.

All canned cycles are performed with respect to the currently selected plane. Any of the three planes (XY, YZ, ZX) may be selected. Throughout this section, most of the descriptions assume the XY-plane has been selected. The behavior is always analogous if the YZ or XZ-plane is selected.

Rotational axis words are allowed in canned cycles, but it is better to omit them. If rotational axis words are used, the numbers must be the same as the current position numbers so that the rotational axes do not move.

All canned cycles use X, Y, R, and Z numbers in the NC code. These numbers are used to determine X, Y, R, and Z positions. The R (usually meaning retract) position is along the axis perpendicular to the currently selected plane (Z-axis for XY-plane, X-axis for YZ-plane, Y-axis for XZ-plane). Some canned cycles use additional arguments.

For canned cycles, we will call a number "sticky" if, when the same cycle is used on several lines of code in a row, the number must be used the first time, but is optional on the rest of the lines. Sticky numbers keep their value on the rest of the lines if they are not explicitly programmed to be different. The R number is always sticky.

In incremental distance mode: when the XY-plane is selected, X, Y, and R numbers are treated as increments to the current position and Z as an increment from the Z-axis position before the move involving Z takes place; when the YZ or XZ-plane is selected,

treatment of the axis words is analogous. In absolute distance mode, the X, Y, R, and Z numbers are absolute positions in the current coordinate system.

The L number is optional and represents the number of repeats. L=0 is not allowed. If the repeat feature is used, it is normally used in incremental distance mode, so that the same sequence of motions is repeated in several equally spaced places along a straight line. In absolute distance mode, L > 1 means "do the same cycle in the same place several times," Omitting the L word is equivalent to specifying L=1. The L number is not sticky.

When L>1 in incremental mode with the XY-plane selected, the X and Y positions are determined by adding the given X and Y numbers either to the current X and Y positions (on the first go-around) or to the X and Y positions at the end of the previous go-around (on the repetitions). The R and Z positions do not change during the repeats.

The height of the retract move at the end of each repeat (called "clear Z" in the descriptions below) is determined by the setting of the retract mode: either to the original Z position (if that is above the R position and the retract mode is G98), or otherwise to the R position.

It is an error if:

- **X, Y, and Z words are all missing during a canned cycle,**
- **a P number is required and a negative P number is used,**
- **an L number is used that does not evaluate to a positive integer,**
- **rotational axis motion is used during a canned cycle,**
- **inverse time feed rate is active during a canned cycle,**
- **cutter radius compensation is active during a canned cycle.**

When the XY plane is active, the Z number is sticky, and it is an error if:

- **the Z number is missing and the same canned cycle was not already active,**
- **the R number is less than the Z number.**

When the XZ plane is active, the Y number is sticky, and it is an error if:

¨ the Y number is missing and the same canned cycle was not already active,

¨ the R number is less than the Y number.

When the YZ plane is active, the X number is sticky, and it is an error if:

¨ the X number is missing and the same canned cycle was not already active,

¨ the R number is less than the X number.

Preliminary and In-Between Motion

At the very beginning of the execution of any of the canned cycles, with the XY-plane selected, if the current Z position is below the R position, the Z-axis is traversed to the R position. This happens only once, regardless of the value of L.

In addition, at the beginning of the first cycle and each repeat, the following one or two moves are made:

a straight traverse parallel to the XY-plane to the given XY-position,

a straight traverse of the Z-axis only to the R position, if it is not already at the R position.

If the XZ or YZ plane is active, the preliminary and in-between motions are analogous.

Return to G-code list

G81 Drill Cycle

The G81 cycle is intended for drilling. Program G81 X~ Y~ Z~ A~ B~ C~ R~ L~

- Preliminary motion, as described above.
- Move the Z-axis only at the current feed rate to the Z position.
- Retract the Z-axis at traverse rate to clear Z.

Example 1. Suppose the current position is (1, 2, 3) and the XY-plane has been selected, and the following line of NC code is interpreted.

G90 G81 G98 X4 Y5 Z1.5 R2.8

This calls for absolute distance mode (G90), old "Z" retract mode (G98) and calls for the G81 drilling cycle to be performed once. The X number and X position are 4. The Y number and Y position are 5. The Z number and Z position are 1.5. The R number and clear Z are 2.8. The following moves take place.

- a traverse parallel to the XY-plane to (4,5,3)
- a traverse parallel to the Z-axis to (4,5,2.8)
- a feed parallel to the Z-axis to (4,5,1.5)
- a traverse parallel to the Z-axis to (4,5,3)

Example 2. Suppose the current position is (1, 2, 3) and the XY-plane has been selected, and the following line of NC code is interpreted.

G91 G81 G98 X4 Y5 Z-0.6 R1.8 L3

This calls for incremental distance mode (G91), old "Z" retract mode and calls for the G81 drilling cycle to be repeated three times. The X number is 4, the Y number is 5, the Z number is -0.6 and the R number is 1.8. The initial X position is 5 (=1+4), the initial Y position is 7 (=2+5), the clear Z position is 4.8 (=1.8+3), and the Z position is 4.2 (=4.8-0.6). Old Z is 3.0

The first move is a traverse along the Z-axis to (1,2,4.8), since old Z < clear Z.

The first repeat consists of 3 moves.

- a traverse parallel to the XY-plane to (5,7,4.8)
- a feed parallel to the Z-axis to (5,7, 4.2)
- a traverse parallel to the Z-axis to (5,7,4.8)

The second repeat consists of 3 moves. The X position is reset to 9 (=5+4) and the Y position to 12 (=7+5).

- a traverse parallel to the XY-plane to (9,12,4.8)
- a feed parallel to the Z-axis to (9,12, 4.2)

- a traverse parallel to the Z-axis to (9,12,4.8)

The third repeat consists of 3 moves. The X position is reset to 13 (=9+4) and the Y position to 17 (=12+5).

- a traverse parallel to the XY-plane to (13,17,4.8)
- a feed parallel to the Z-axis to (13,17, 4.2)
- a traverse parallel to the Z-axis to (13,17,4.8)

Return to G-code list

G82 Drill Cycle with Dwell

The G82 cycle is intended for drilling. Program

G82 X~ Y~ Z~ A~ B~ C~ R~ L~ P~

- Preliminary motion, as described above.
- Move the Z-axis only at the current feed rate to the Z position.
- Dwell for the P number of seconds.
- Retract the Z-axis at traverse rate to clear Z.

Return to G-code list

G83 Peck Drill Cycle

The G83 cycle (often called peck drilling) is intended for deep drilling or milling with chip breaking. See also G73. The retracts in this cycle clear the hole of chips and cut off any long stringers (which are common when drilling in aluminum). This cycle takes a Q number which represents a "delta" increment along the Z-axis. Program

G83 X~ Y~ Z~ A~ B~ C~ R~ L~ Q~

- Preliminary motion, as described above.

- Move the Z-axis only at the current feed rate downward by delta or to the Z position, whichever is less deep.
- Rapid back out to the clear Z.
- Rapid back down to the current hole bottom, backed off a bit.
- Repeat steps 1, 2, and 3 until the Z position is reached at step 1.
- Retract the Z-axis at traverse rate to clear Z.

It is an error if:

- the Q number is negative or zero.

Return to G-code list

G84 Not Supported

This code is dependant on the motion device used.

Return to G-code list

G85 Boring or Reaming Cycle

The G85 cycle is intended for boring or reaming, but could be used for drilling or milling.

Program G85 X~ Y~ Z~ A~ B~ C~ R~ L~

- Preliminary motion, as described above.
- Move the Z-axis only at the current feed rate to the Z position.
- Retract the Z-axis at the current feed rate to clear Z.

Return to G-code list

G86 Boring Cycle

The G86 cycle is intended for boring. This cycle uses a P number for the number of seconds to dwell. Program G86 X~ Y~ Z~ A~ B~ C~ R~ L~ P~

- **Preliminary motion, as described above.**
- **Move the Z-axis only at the current feed rate to the Z position.**
- **Dwell for the P number of seconds.**
- **Stop the spindle turning.**
- **Retract the Z-axis at traverse rate to clear Z.**
- **Restart the spindle in the direction it was going.**

The spindle must be turning before this cycle is used. It is an error if:

- **the spindle is not turning before this cycle is executed.**

Return to G-code list

G87 Cycle

This code is dependant on the motion device used.

Return to G-code list

G88 Boring Cycle

The G88 cycle is intended for boring. This cycle uses a P word, where P specifies the number of seconds to dwell. Program G88 X~ Y~ Z~ A- B~ C~ R~~ L~ P~

- **Preliminary motion, as described above.**
- **Move the Z-axis only at the current feed rate to the Z position.**
- **Dwell for the P number of seconds.**
- **Stop the spindle turning.**

- **Stop the program so the operator can retract the spindle manually.**
- **Restart the spindle in the direction it was going.**

Return to G-code list

G89 Boring Cycle

The G89 cycle is intended for boring. This cycle uses a P number, where P specifies the number of seconds to dwell. program G89 X~ Y~ Z~ A~ B~ C~ R~ L~ P~

- **Preliminary motion, as described above.**
- **Move the Z-axis only at the current feed rate to the Z position.**
- **Dwell for the P number of seconds.**
- **Retract the Z-axis at the current feed rate to clear Z.**

Return to G-code list

G90 & G91 Distance Mode

Interpretation of Mac code can be in one of two distance modes: absolute or incremental.

To go into absolute distance mode, program G90. In absolute distance mode, axis numbers (X, Y, Z, A, B, C) usually represent positions in terms of the currently active coordinate system. Any exceptions to that rule are described explicitly in this section describing G-codes.

To go into incremental distance mode, program G91. In incremental distance mode, axis numbers (X, Y, Z, A, B, C) usually represent increments from the current values of the numbers.

I and J numbers always represent increments, regardless of the distance mode setting. K numbers represent increments in all but one usage (the G87 boring cycle), where the meaning changes with distance mode.

Return to G-code list

G90.1 & G91.1 Set IJK Arc Mode

Interpretation of the IJK values in G02 and G03 codes can be in one of two distance modes: absolute or incremental.

To go into absolute IJ mode, program G90.1. In absolute distance mode, IJK numbers represent absolute positions in terms of the currently active coordinate system.

To go into incremental IJ mode, program G91.1. In incremental distance mode, IJK numbers usually represent increments from the current controlled point.

Incorrect settings of this mode will generally result in large incorrectly oriented arcs in the toolpath display.

Return to G-code list

G92,G92.1,G92.2 & G92.3 Offsets

See the chapter on coordinate systems for full details. You are strongly advised not to use this legacy feature on any axis where there is another offset applied.

To make the current point have the coordinates you want (without motion), program G92 X~ Y~ Z~ A~ B~ C~ , where the axis words contain the axis numbers you want. All axis words are optional, except that at least one must be used. If an axis word is not used for a given axis, the coordinate on that axis of the current point is not changed. It is an error if:

- all axis words are omitted.

G52 and G92 use common internal mechanisms in Mac and may not be used together.

When G92 is executed, the origin of the currently active coordinate system moves. To do this, origin offsets are calculated so that the coordinates of the current point with respect to the moved origin are as specified on the line containing the G92. In addition, parameters 5211 to 5216 are set to the X, Y, Z, A, B, and C-axis offsets. The offset for an axis is the amount the origin must be moved so that the coordinate of the controlled point on the axis has the specified value.

Here is an example. Suppose the current point is at X=4 in the currently specified coordinate system and the current X-axis offset is zero, then G92 X7 sets the X-axis offset to -3, sets parameter 5211 to -3, and causes the X-coordinate of the current point to be 7.

The axis offsets are always used when motion is specified in absolute distance mode using any of the fixture coordinate systems Thus all fixture coordinate systems are affected by G92.

Being in incremental distance mode has no effect on the action of G92.

Non-zero offsets may be already be in effect when the G92 is called. They are in effect discarded before the new value is applied. Mathematically the new value of each offset is A+B, where A is what the offset would be if the old offset were zero, and B is the old offset. For example, after the previous example, the X-value of the current point is 7. If G92 X9 is then programmed, the new X-axis offset is -5, which is calculated by [[7-9] + -3]. Put another way the G92 X9 produces the same offset whatever G92 offset was already in place.

To reset axis offsets to zero, program G92.1 or G92.2 G92.1 sets parameters 5211 to 5216 to zero, whereas G92.2 leaves their current values alone.

To set the axis offset values to the values given in parameters 5211 to 5216, program G92.3

You can set axis offsets in one program and use the same offsets in another program. Program G92 in the first program. This will set parameters 5211 to 5216. Do not use G92.1 in the remainder of the first program. The parameter values will be saved when the first program exits and restored when the second one starts up. Use G92.3 near the beginning of the second program. That will restore the offsets saved in the first program.

Return to G-code list

G93 Inverse Time

In inverse time feed rate mode, an F word means the move should be completed in [one divided by the F number] minutes. For example, if the F number is 2.0, the move should be completed in half a minute.

Return to G-code list

G94 Units Per Minute

In units per minute feed rate mode, an F word on the line is interpreted to mean the controlled point should move at a certain number of inches per minute, millimetres per minute, or degrees per minute, depending upon what length units are being used and which axis or axes are moving.

Return to G-code list

G98 & G99 Canned Cycle Return

When the spindle retracts during canned cycles, there is a choice of how far it retracts:

- **retract perpendicular to the selected plane to the position indicated by the R word, or**
- **retract perpendicular to the selected plane to the position that axis was in just before the canned cycle started (unless that position is lower than the position indicated by the R word, in which case use the R word position).**

To use option (1), program G99 To use option (2), program G98 Remember that the R word has different meanings in absolute distance mode and incremental distance mode.