

PARTIAL G CODE COMMAND SET

The following is only a partial Mx4 NC Executive command set:

G CODE SUMMARY

G00	Rapid Motion
G01	Linear Motion
G02	Clockwise Circular Motion
G03	Counter-Clockwise Circular Motion
G04	Dwell
G09	Exact Stop Check
G17	Use XY Plane for Circular Motion
G18	Use ZX Plane for Circular Motion
G19	Use YZ Plane for Circular Motion
G20	Millimeter Mode
G21	Inch Mode
G28	Return to Reference Point
G29	Set Return Reference Point
G40	Cutter Compensation Cancel
G41	Cutter Compensation Left
G42	Cutter Compensation Right
G43	Tool Length Compensation +
G44	Tool Length Compensation -
G45	Tool Offset Increase
G46	Tool Offset Decrease
G47	Tool Offset Doubled Increase
G48	Tool Offset Doubled Decrease
G49	Cancel G43 and G44
G52	Set Local Coordinate System
G54	Select Work Coordinate System 1
G55	Select Work Coordinate System 2
G56	Select Work Coordinate System 3

Mx4 GCode Command Set

G57	Select Work Coordinate System 4
G58	Select Work Coordinate System 5
G59	Select Work Coordinate System 6
G60	Uni-Directional Positioning
G61	Exact Stop Modal
G70-72	Bolt Hole Circle, ARC and Along an Angle
G90	Absolute Positioning
G91	Incremental Positioning
G92	Define Relative Origin

M CODE SUMMARY

M00	Stop Program
M01	Optional Program Stop
M02	Program End
M03	Spindle Forward (Clockwise)
M04	Spindle Reverse (Counter-Clockwise)
M05	Spindle Stop
M06	Tool Change
M60-91	Output On
M92-123	Output Off

G AND M CODE LISTINGS

The following notation will be used:

C Auxiliary external rotary axis

The C address character is used to specify motion for the optional external axis C. It specifies an angle in degrees for the rotary axis. It is always followed by a signed number and up to three fractional decimal positions.

D Tool diameter selection

The D address character is used to select the tool diameter or radius used for cutter compensation. The number following must be between 0 and 50.

F Feed rate

The F address character is used to select the feed rate applied to any interpolation functions. It is either in inches per minute with four fractional positions or mm per minute with three fractional positions.

H Tool length offset selection

The H address character is used to select the tool length offset entry from the offsets memory. The number following must be between 0 and 50.

I Center distance from the starting point of the arc in x direction

It is either in inches with four fractional positions or mm with three fractional positions.

Mx4 GCode Command Set

- J** Center distance from the starting point of the arc in y direction
- It is either in inches with four fractional positions or mm with three fractional positions.
- P** Delay time
- The P address character is used to enter a time in seconds (for a G04 dwell).
- S** Spindle speed
- Indicates the percentage of peak speed (1-100).
- T** Tool selection code
- The T address character is used to select the tool for the next tool change. The number following must be between 1 and 16.
- X** Linear X-axis motion
- The X address character is used to specify motion for the X-axis. It specifies a position or distance along the X-axis. It is either inches with four fractional positions or mm with three fractional positions.
- Y** Linear Y-axis motion
- The Y address character is used to specify motion for the Y-axis. It specifies a position or distance along the Y-axis. It is either inches with four fractional positions or mm with three fractional positions.
- Z** Linear Z-axis motion
- The Z address character is used to specify motion for the Z-axis. It specifies a position or distance along the Z-axis. It is either inches with four fractional positions or mm with three fractional positions.

GOO RAPID MOTION

X	Motion command in x direction (Optional)
Y	Motion command in y direction (Optional)
Z	Motion command in z direction (Optional)

This G code provides a rapid traverse of three axes of the machine. The auxiliary C (rotary axis) can also be moved by this command. This G code is modal so that a previous block with G00 causes all following blocks to be rapid motions until another group (e.g. G01) is specified. The rapid traverse rate is dependent on the maximum speed specified for each axis (g_dspl.par).

Generally, rapid motions will not be in straight lines. All of the axes specified are moved at the same time but will not necessarily complete their motions at the same time. The block will wait until all motions are complete.

GO1 LINEAR MOTION

F	Feed rate in inches (mm) per minute
X	Motion command in x direction (Optional)
Y	Motion command in y direction (Optional)
Z	Motion command in z direction (Optional)

This G code provides straight linear motion from a (one to three dimensional) point to another. All axes start and stop at the same time, whereas with G00 although axes start simultaneously they don't necessarily finish together.

The rotary axis, C, may be commanded by G01. When C is commanded the resulting motion trajectory will be helical. The speed of all axes are specified by feed rate (F command) so that the helical path will have consistent speed. Rotary axis speed depends on rotary axis diameter setting.

GO2 CLOCKWISE CIRCULAR MOTION

Mx4 GCode Command Set

F	Feed rate in inches (mm) per minute
I	Distance along x axis to center of circle (Optional)
J	Distance along y axis to center of circle (Optional)
K	Distance along z axis to center of circle (Optional)
X	Motion command in x direction (Optional)
Y	Motion command in y direction (Optional)
Z	Motion command in z direction (Optional)

This G code provides clockwise circular motion of two of the axes. The G codes G17, G18 and G19 determine the circular planes in XY, ZX or YZ directions respectively. The X, Y and Z are used to specify the end point of the motion. If arguments for selected X, Y or Z are not specified the end point of the arc will be the same as the starting point for that axis. The center for the arc in X, Y or Z directions is specified by I, J or K respectively. These values specify the distant form the center of the arc to the starting point of the arc.

GO3 COUNTER-CLOCKWISE CIRCULAR MOTION

F	Feed rate in inches (mm) per minute
I	Distance along x axis to center of circle (Optional)
J	Distance along y axis to center of circle (Optional)
K	Distance along z axis to center of circle (Optional)
X	Motion command in x direction (Optional)
Y	Motion command in y direction (Optional)
Z	Motion command in z direction (Optional)

This G code provides counter-clockwise circular motion of two of the axes. The G codes G17, G18 and G19 determine the circular planes in XY, ZX or YZ directions respectively. The X, Y and Z are used to specify the end point of the motion. If arguments for selected X, Y or Z are not specified the end point of the arc will be the same as the starting point for that axis. The center for the arc in X, Y or Z directions is specified by I, J or K respectively. These values specify the distant form the center of the arc to the starting point of the arc.

GO4 DWELL

P	The dwell time in seconds
---	---------------------------

This G code provides delay in the program. The block containing G04 will delay for the time specified in the P code.

G09 EXACT STOP CHECK

This G code provides exact stop. Using this code, makes rapid and interpolated moves decelerate to a stop before the next block is processed. This command is non-modal which means the following blocks are not affected by it. In exact stop, moves take longer time when compared to continuous cutter motion.

G17 USE XY PLANE FOR CIRCULAR MOTION

This mode is modal and applies to all following circular motions (see G02 and G03).

G18 USE ZX PLANE FOR CIRCULAR MOTION

This mode is modal and applies to all following circular motions (see G02 and G03).

G19 USE YZ PLANE FOR CIRCULAR MOTION

This mode is modal and applies to all following circular motions (see G02 and G03).

G20 MILLIMETER MODE

G20 sets the position unit for all subsequent G code lines to millimeter mode. For example, X3.0 is interpreted as 3 mm.

G21 INCH MODE

G21 sets the position unit for all subsequent G code lines to inch mode. For example, X3.0 is interpreted as 3 inches.

G28 RETURN TO REFERENCE POINT

The G28 code is used to return to the machine zero position on all axes. If an X, Y, or Z code is specified on the same block, only those axes will be moved and they will be moved to the specified positions in the current coordinate system and then they will be moved to machine zero. The intermediate position, if specified, is saved for use in the G29. If no X, Y, or Z is specified, all axes will be moved directly to machine zero.

G29 SET RETURN REFERENCE POINT

The G29 code is used to move the axes to a position via a previously –set reference point. The reference is defined with the G29. This command is normally given with the axes positioned at machine zero. The axes which are selected in this block are moved first to the intermediate reference point and then they are moved to the X, Y, or Z specified. The positions are interpreted in the current coordinate system.

G40 CUTTER COMPENSATION CANCEL

G40 cancels cutter compensation G41 and G42.

G41 CUTTER COMPENSATION LEFT

This G code selects cutter compensation left. That is, the tool is moved to the left of the programmed path to compensate for the diameter of the tool. A Dxx (where xx is an integer value) must be programmed to select the tool size from the setup file.

G42 CUTTER COMPENSATION RIGHT

This G code selects cutter compensation right. That is, the tool is moved to the right of the programmed path to compensate for the diameter of the tool. A Dxx

(where xx is an integer value) must be programmed to select the tool size from the setup file.

G43 TOOL LENGTH COMPENSATION +(PLUS)

This G code selects tool length compensation in a positive direction. That is, the tool length offset is added to the Z axis position command. A non-zero Hxx (where xx is an integer value) must be programmed to select the proper offset from the setup file.

G44 TOOL LENGTH COMPENSATION -(MINUS)

This G code selects tool length compensation in negative direction. That is, the tool length offset is subtracted from the Z axis position command. A non-zero Hxx (where xx is an integer value) must be programmed to select the proper offset from the setup file.

G49 CANCEL G43 / G44

This G code cancels the tool length compensation. H0 will also cancel tool length compensation.

G52 SET LOCAL COORDINATE SYSTEM

This G code selects the work coordinate systems which can be set with G92.

G54-59 SELECT WORK COORDINATE SYSTEM 1-6

Mx4 GCode Command Set

These codes select one of the six user coordinate systems stored within the offsets memory. All subsequent references to axes positions will be interpreted in the new coordinate system.

G60 UNI-DIRECTIONAL POSITIONING

This G code is used to provide positioning always from the plus direction. It is non-modal so does not effect the following blocks.

G61 EXACT STOP MODAL

Rapid and interpolated moves will decelerate to an exact stop before another block is processed. It is modal and therefore it affects the following blocks.

G90 ABSOLUTE POSITIONING

This G code enforces absolute positioning in the user's selected coordinate. It is modal, thus affects the following blocks until otherwise (G91) specified. Each axis which is moved will be placed at the position coded in the command block.

G91 INCREMENTAL POSITIONING

This G code enforces incremental positioning in the user's selected coordinate. It is modal, thus affects the following blocks until otherwise (G90) specified. Each axis which is moved will be moved by the amount coded in the command block.

G92 DEFINE RELATIVE ORIGIN

This command does not move any of the axes; it only changes the values stored as user work offset. Yasnac style of coordinates: all of the values of X, Y, and Z in this block are stored into the offset entry for G52. Fanuc style of coordinates: uses G92 to change all the user work coordinates.

M00 STOP PROGRAM

This code is used to stop the program. It also stops the spindle. The program pointer will advance to the next block and stop. A cycle start will continue program operation from the next block.

M01 OPTIONAL PROGRAM STOP

Same as M00, except that it works only if the OPTIONAL STOP is turned on (user may set the DSPL variable defined in filename.hll as Optional_Stop)

M02 PROGRAM END

Same as M00, except program pointer is not advanced to the next block.

M03 SPINDLE FORWARD (CLOCKWISE)

Starts spindle in clockwise direction at the speed set in g_dspl.par. The block is delayed until spindle reaches 90% commanded speed. This M code can be used in conjunction with the S parameter which will allow the speed of the spindle to be programmed as a % of peak speed.

M04 SPINDLE REVERSE (COUNTER-CLOCKWISE)

Starts spindle in counter- clockwise direction at the speed set in g_dspl.par. The block is delayed until spindle reaches 90% of commanded speed. This M code can be used in conjunction with the S parameter which will allow the speed of the spindle to be programmed as a % of peak speed.

M05 SPINDLE STOP

This code is used to stop the spindle. The block is delayed until the spindle slows below 10 RPM.

M06 TOOL CHANGE (GENERIC PLC PROGRAM)

This M code implements tool change as specified by Tnn where nn is the tool number. Essentially, a generic PLC subroutine specified by the user can be called based on the tool number. See *Mx4 GCode Programming: M06 tool change DSPL file [tool_chg.hll]*.

M60-M91 OUTPUT ON

Turn Mx4 controller discrete output on. M60-M91 correspond to outputs OUTP1-OUTP32, respectively. For example, M62 turns on output OUTP3. See *Mx4 User's Guide* chapter 2 for hardware information on Mx4's outputs.

M92-M123 OUTPUT OFF

Turn Mx4 controller discrete output off. M92-M123 correspond to outputs OUTP1-OUTP32, respectively. For example, M92 turns off output OUTP1. See *Mx4 User's Guide* chapter 2 for hardware information on Mx4's outputs.