



Onefinity CNC Controller v1.0.9

GCode Cheat Sheet

Code	Parameters	Description	
-------------	-------------------	--------------------	--

Motion

G0		Rapid Move	
G1		Linear Move	
G2, G3	I J K or R, P	Arc Move	
G4	P	Dwell	
G5	I J P Q	Cubic Spline	(not supported)
G5.1	I J	Quadratic Spline	(not supported)
G5.2	P L	NURBS	(not supported)
G33.1	K	Rigid Tapping	(not supported)

Homing & Probing

G28.2, G28.3		(Un)set Axis Homed State	
G38.2 - G38.5		Straight Probe	
G38.6 - G38.9		Seek Switch	

Tool Control

I		Select Tool	
M6	T	Tool Change	
M61	Q	Set Current Tool	(not supported)
G10 L1	P Q R	Set Tool Table	
G10 L10	P	Set Tool Table	
G10 L11	P	Set Tool Table	
G43	H	Tool Length Offset	
G43.1		Dynamic Tool Length Offset	
G43.2	H	Apply additional Tool Length Offset	
G49		Cancel Tool Length Compensation	

Feed Control

F		Set Feed Rate	
-------------------	--	---------------	--

G93, G94, G95		Feed Rate Mode	(not supported)
M52	P0 (off) or P1 (on)	Adaptive Feed Control	(not supported)
M53	P0 (off) or P1 (on)	Feed Stop Control	(not supported)

Spindle Control

S		Set Spindle Speed	
M3, M4, M5	S	Spindle Control	
M19		Orient Spindle	(not supported)
G96, G97	S D	Spindle Control Mode	(not supported)
G33	K	Spindle Synchronized Motion	(not supported)

Coolant

M7, M8, M9		Coolant Control	
----------------------------	--	-----------------	--

Stopping

M0, M1		Program Pause	
M2, M30		Program End	
M60		Pallet Change Pause	(not supported)

Units

G20, G21		Units (inch, mm)	
--------------------------	--	------------------	--

Distance Mode

G90, G91		Distance Mode	
G90.1, G91.1		Arc Distance Mode	
G7		Lathe Diameter Mode	(not supported)
G8		Lathe Radius Mode	(not supported)

Cutter Radius Compensation

			(not supported)
G40		Compensation Off	(not supported)
G41, G42	D	Cutter Compensation	(not supported)
G41.1, G42.1	D L	Dynamic Cutter Compensation	(not supported)

Path Control Mode

G61 G61.1		Exact Path Mode	
G64	P Q	Path Blending (Partial support)	

Overrides

M48, M49		Speed and Feed Override Control	(not supported)
M50	P0 (off) or P1 (on)	Feed Override Control	(not supported)
M51	P0 (off) or P1 (on)	Spindle Speed Override Control	(not supported)

Coordinate Systems, Offsets & Planes

G54-G59.3		Select Coordinate System	
G10 L2	P R	Set Coordinate System	
G10 L20	P	Set Coordinate System	
G52		Local Coordinate System Offset	
G53		Move in Machine Coordinates	
G92		Coordinate System Offset	
G92.1, G92.2		Reset G92 Offsets	
G92.3		Restore G92 Offsets	
G28, G28.1		Go/Set Predefined Position	
G30, G30.1		Go/Set Predefined Position	
G17 - G19.1	(affects G2, G3, G81...G89, G40...G42)	Plane Select	

Flow-control Codes

o sub/endsub/call		Subroutines, sub/endsub call	
o while		Looping, while/endwhile do/while	
o if		Conditional, if/else/endif	
o repeat		Repeat a loop of code	
[]		Indirection	
o call		Call named or numbered file	

Modal State

M70		Save modal state	
M71		Invalidate stored state	
M72		Restore modal state	

M73		Save and Auto-restore modal state	
---------------------	--	-----------------------------------	--

Input/Output

M62 - M65	P	Digital Output Control	(not supported)
M66	P E L Q	Wait on Input	(not supported)
M67	T	Analog Output,Synchronized	(not supported)
M68	T	Analog Output, Immediate	(not supported)

User Defined Commands

M101 - M199	P Q	User Defined Commands	(not supported)
-----------------------------	-----	-----------------------	-----------------

Canned cycles

G80		Cancel Canned Cycle	
G81	R L (P)	Drilling Cycle	
G82	R L (P)	Drilling Cycle, Dwell	
G83	R L Q	Drilling Cycle, Peck	(not supported)
G73	R L Q	Drilling Cycle, Chip Breaking	
G85	R L (P)	Boring Cycle, Feed Out	
G89	R L (P)	Boring Cycle, Dwell, Feed Out	
G76	P Z I J R K Q H L E	Threading Cycle	(not supported)
G98, G99		Canned Cycle Return Level	(not supported)

Comments & Messages

; (...)		Comments	
(MSG,...)		Messages	
(DEBUG,...)		Debug Messages	
(PRINT,...)		Print Messages	(not supported)
(LOG,...)		Logging Messages	

Further GCode Programming Documentation

The Buildbotics controller implements a subset of LinuxCNC GCode. Supported commands are listed above. You can find further help with [GCode](#) programming on the LinuxCNC website:

- [G Code overview](#)
- [G Code reference](#)
- [M Code reference](#)