PREPARATORY (G-CODES) & MISCELLANEOUS (M-CODES)

G-CODES REFERENCE

A complete listing of **G-Codes** with definitions for both turning and milling

Milling		Turning	
G00	Positioning in Rapid	G00	Positioning in Rapid
G01	Linear Interpolation	G01	Linear Interpolation
G02	Circular Interpolation (CW)	G02	Circular Interpolation (CW)
G03	Circular Interpolation (CCW)	G03	Circular Interpolation (CCW)
G04	Dwell	G04	Dwell
G07	Imaginary axis designation	G07	Feedrate sine curve control
G09	Exact stop check	.	.
G10	Program parameter input	G10	Data setting
G11	Program parameter input cancel	G11	Data setting cancel
G12	Circle Cutting CW	.	.
G13	Circle Cutting CCW	.	.
G17	XY Plane	G17	XY Plane
G18	XZ Plane	G18	XZ Plane
G19	YZ Plane	G19	YZ Plane
G20	Inch Units	G20	Inch Units
G21	Metric Units	G21	Metric Units
G22	Stored stroke limit ON	G22	Stored stroke check function ON
G23	Stored stroke limit OFF	G23	Stored stroke check function OFF
.	.	G25	Spindle speed fluctuation detection OFF
.	.	G26	Spindle speed fluctuation detection ON
G27	Reference point return check	G27	Reference point return check
G28	Automatic return to reference point	G28	Automatic Zero Return
G29	Automatic return from reference point	G29	Return from Zero Return Position
G30	Return to 2nd, 3rd, 4th reference point	G30	2nd reference point return
G31	Skip function	G31	Skip function
.	.	G32	Thread cutting
G33	Thread cutting	.	.
G34	Bolt hole circle (Canned Cycle)	G34	Variable lead thread cutting
G35	Line at angle (Canned Cycle)	.	.
G36	Arc (Canned Cycle)	G36	Automatic tool compensation
G40	Cutter compensation Cancel	G40	Tool Nose Radius Compensation Cancel
G41	Cutter compensation Left	G41	Tool Nose Radius Compensation Left
G42	Cutter compensation Right	G42	Tool Nose Radius Compensation Right
G43	Tool Length Compensation (Plus)	.	.

G44	Tool Length Compensation (Minus)	.	.
G45	Tool offset increase	.	.
G46	Tool offset decrease	G46	Automatic Tool Nose Radius Compensation
G47	Tool offset double increase	.	.
G48	Tool offset double decrease	.	.
G49	Tool Length Compensation Cancel	.	.
G50	Scaling OFF	G50	Coordinate system setting and maximum rpm
G51	Scaling ON	.	.
G52	Local coordinate system setting	G52	Local coordinate system setting
G53	Machine coordinate system selection	G53	Machine coordinate system setting
G54	Work piece Coordinate System	G54	Work piece Coordinate System
G55	Work piece Coordinate System 2	G55	Work piece Coordinate System 2
G56	Work piece Coordinate System 3	G56	Work piece Coordinate System 3
G57	Work piece Coordinate System 4	G57	Work piece Coordinate System 4
G58	Work piece Coordinate System 5	G58	Work piece Coordinate System 5
G59	Work piece Coordinate System 6	G59	Work piece Coordinate System 6
G60	Single direction positioning	.	.
G61	Exact stop check mode	G61	Exact stop check mode
G62	Automatic corner override	G62	Automatic corner override
G63	Tapping mode	G63	Tapping mode
G64	Cutting mode	G64	Cutting mode
G65	Custom macro simple call	G65	User macro simple call
G66	Custom macro modal call	G66	User macro modal call
G67	Custom macro modal call cancel	G67	User macro modal call cancel
G68	Coordinate system rotation ON	G68	Mirror image for double turrets ON
G69	Coordinate system rotation OFF	G69	Mirror image for double turrets OFF
G70	Inch Units	G70	Finishing Cycle
G71	Metric Units	G71	Turning Cycle
G72	User canned cycle	G72	Facing Cycle
G73	High-Speed Peck Drilling Cycle	G73	Pattern repeating
G74	Counter tapping cycle	G74	Peck Drilling Cycle
G75	User canned cycle	G75	Grooving Cycle
G76	Fine boring cycle	G76	Threading Cycle
G77	User canned cycle	.	.
G78	User canned cycle	.	.
G79	User canned cycle	.	.
G80	Cancel Canned Cycles	G80	Canned cycle for drilling cancel
G81	Drilling Cycle	.	.
G82	Counter Boring Cycle	.	.
G83	Deep Hole Drilling Cycle	G83	Face Drilling Cycle
G84	Tapping cycle	G84	Face Tapping Cycle
G85	Boring Cycle	.	.

G86	Boring Cycle	G86	Face Boring Cycle
G87	Back Boring Cycle	G87	Side Drilling Cycle
G88	Boring Cycle	G88	Side Tapping Cycle
G89	Boring Cycle	G89	Side Boring Cycle
G90	Absolute Positioning	G90	Absolute Programming
G91	Incremental Positioning	G91	Incremental Programming
G92	Reposition Origin Point	G92	Thread Cutting Cycle
G93	Inverse time feed	.	.
G94	Per minute feed	G94	End face Turning Cycle
G95	Per revolution feed	.	.
G96	Constant surface speed control	G96	Constant surface speed control
G97	Constant surface speed control cancel	G97	Constant surface speed control cancel
G98	Set Initial Plane default	G98	Linear Feedrate Per Time
G99	Return to Retract (Rapid) Plane	G99	Feedrate Per Revolution
.	.	G107	Cylindrical Interpolation
.	.	G112	Polar coordinate interpolation mode
.	.	G113	Polar coordinate interpolation mode cancel
.	.	G250	Polygonal turning mode cancel
.	.	G251	Polygonal turning mode

M-CODES REFERENCE

A complete listing of M-Codes with definitions for both turning and milling

Milling		Turning	
M00	Program Stop	M00	Program Stop
M01	Optional Program Stop	M01	Optional Program Stop
M02	Program End	M02	Program End
M03	Spindle On Clockwise	M03	Spindle On Clockwise
M04	Spindle On Counter clockwise	M04	Spindle On Counter clockwise
M05	Spindle Stop	M05	Spindle Stop
M06	Tool Change	.	.
.	.	M07	Coolant 1 On
M08	Coolant On	M08	Coolant 2 On
M09	Coolant Off	M09	Coolant Off
M10	Clamps On	.	.
M11	Clamps Off	.	.
M30	End of Program, Reset to Start	M30	End of Program, Reset to Start
M98	Call subroutine command	M98	Subprogram call
M99	Return from subroutine command	M99	Return from subprogram

LETTER ADDRESSES

Some letter addresses are used only in milling or only in turning; most are used in both.

Variable	Description
A	Absolute or incremental position of A axis (rotational axis around X axis)
B	Absolute or incremental position of B axis (rotational axis around Y axis)
C	Absolute or incremental position of C axis (rotational axis around Z axis)
D	Defines diameter or radial offset used for cutter compensation. D is used for depth of cut on lathes. It is used for aperture selection and commands on photo plotters.
E	Precision feedrate for threading on lathes
F	Defines feed rate
G	Address for preparatory commands
H	Defines tool length offset;
	Incremental axis corresponding to C axis (e.g., on a turn-mill)
I	Defines arc center in X axis for G02 or G03arc commands.
	Also used as a parameter within some fixed cycles.
J	Defines arc center in Y axis for G02 or G03arc commands.
	Also used as a parameter within some fixed cycles.
K	Defines arc center in Z axis for G02 or G03arc commands.
	Also used as a parameter within some fixed cycles, equal to L address.
L	Fixed cycle loop count;

	Specification of what register to edit usingG10
M	Miscellaneous function
N	Line (block) number in program;
	System parameter number to be changed using G10
O	Program name
P	Serves as parameter address for various G and M codes
Q	Peck increment in canned cycles
R	Defines size of arc radius, or defines retract height in milling canned cycles
S	Defines speed, either spindle speed or surface speed depending on mode
T	Tool selection
U	Incremental axis corresponding to X axis (typically only lathe group A controls)
	Also defines dwell time on some machines (instead of "P" or "X").
V	Incremental axis corresponding to Y axis
W	Incremental axis corresponding to Z axis (typically only lathe group A controls)
X	Absolute or incremental position of X axis.
	Also defines dwell time on some machines (instead of "P" or "U").
Y	Absolute or incremental position of Y axis
Z	Absolute or incremental position of Z axis