PREPARATORY FUNCTION WORDS

Multiple G words can be included in the same part program block with the following limitations: one G word from each modal group and one non-modal G word (see the chart below). These G words will be discussed in this or following chapters.

G Code	Group	Description	Modal or Non-Modal	Page
*G0	1	Rapid Traverse	Modal	5-5
G1	1	Linear Interpolation	Modal	7-4
G2	1	Circular Interpolation Clockwise (CW)	Modal	7-9
G3	1	Circular Interpolation Counterclockwise (CCW)	Modal	7-9
G4	0	Dwell time	Non-Modal	7-26
G9	0	Exact stop	Non-Modal	7-3
G10	0	Data setting	Non-Modal	
*G17	2	Plane selection XY	Modal	7-24
G18	2	Plane selection ZX	Modal	7-24
G19	2	Plane selection YZ	Modal	7-24
G20	6	Inch programming	Modal	3-2
G21	6	Metric programming	Modal	3-2
G28	0	Reference point return	Non-Modal	5-6
*G40	7	Cancel cutter radius compensation	Modal	8-4
G41	7	Activate cutter radius compensation left	Modal	8-3
G42	7	Activate cutter radius compensation right	Modal	8-3
G43	8	Tool length compensation plus	Modal	5-6
G44	8	Tool length compensation minus	Modal	5-6
G45	0	Tool offset increase	Non-Modal	
G46	0	Tool offset decrease	Non-Modal	
G47	0	Tool offset double increase	Non-Modal	
G48	0	Tool offset double decrease	Non-Modal	
*G49	8	Tool offset cancel	Modal	5-6
G52		Local coordinate system	Modal	12-10
G53	0	Machine coordinate system selection	Non-Modal	12-4
*G54	12	Work coordinate system 1 selection	Modal	5-7
G55	12	Work coordinate system 2 selection	Modal	5-7
G56	12	Work coordinate system 3 selection	Modal	5-7
G57	12	Work coordinate system 4 selection	Modal	5-7
G58	12	Work coordinate system 5 selection	Modal	5-7
G59	12	Work coordinate system 6 selection	Modal	5-7
G60	0	Single direction positioning	Non-Modal	5-5
G61	13	Exact stop mode	Modal	7-2
G62	13	Automatic corner override	Modal	
G63	13	Tapping mode	Modal	
*G64	13	Cutting mode	Modal	7-2
G65	0	Macro call	Non-Modal	11-8
G66	14	Macro call A	Modal	11-12
G66.1	14	Macro call B	Modal	11-14
*G67	14	Macro call A/B cancel	Modal	11-12
G73	9	Peck drilling cycle	Modal	5-24

G Code	Group	Description	Modal or Non-Modal	Page
G74	9	Left hand tap cycle	Modal	5-32
G76	9	Fine bore cycle, spindle orient and axis shift	Modal	5-38
*G80	9	Cancel canned cycle	Modal	5-6
G81	9	Standard drilling cycle	Modal	5-12
G82	9	Drilling cycle with dwell (spot facing)	Modal	5-18
G83	9	Deep hole drilling cycle	Modal	5-21
G84	9	Tapping cycle (servo)	Modal	5-31
G85	9	Boring cycle, feedrate return	Modal	5-34
G86	9	Boring cycle w/ spindle stop, traverse return	Modal	5-36
G87	9	Back boring cycle	Modal	5-41
G89	9	Boring cycle w/ dwell, feedrate return	Modal	5-44
*G90	3	Absolute dimension programming	Modal	5-3
G91	3	Incremental dimension programming	Modal	5-4
G92	0	Preset registers	Non-Modal	12-5
G93	5	Inverse time feedrate programming (1/T)	Modal	15-10
*G94	5	Feed per minute (IPM or MMPM) feedrate programming	Modal	15-10
*G98	10	Canned cycle initial level return	Modal	5-14
G99	10	Canned cycle R point level return	Modal	5-12
G184		Tapping cycle (air) G67 cancels	Modal	5-27
G185		Fast air tapping cycle	Modal	5-29
G198		Fast air tapping cycle selected R point return	Non-Modal	5-30

M0	program stop
M1	optional stop
M2	end of program
M3	spindle start clockwise
M4	spindle start counterclockwise
* M5	spindle stop
M6	tool change
M7	mist coolant on
M8	flood coolant on
* M9	coolant off
M10	high pressure tapping
M12	tool pocket reset
M15	Erickson indexer
M18 (optional)	coolant through spindle
M19	spindle orientation
M20	tool setup (non-ANSI machines only)
M21	tool pocket preset (ANSI machines only)
M24, M25	4th axis clamps off/on (optional)
M26, M27	5th axis clamps off/on (optional)
M28	absolute tool number request
M29	machine lubrication cycle
M30	end of program with return to program start
M31	increment parts count
M32	reset cycle time
M34	reset parts count, lot parts count
M37	set lot parts count
M39	180 spindle orient (machines with optional spindle probe only)
*M48, M49	feedrate override
M66	master tool set
M68	activate spindle probe (machines with optional spindle probe only)
M69	activate TLI (machines with optional TLI only)
* M71	milling clamps off
M72	milling clamps on
M80	non-buffering M code
M395	Hiperdex Zero Set
M396	incremental Hiperdex
*M397	absolute Hiperdex
M98	subprogram call
M99	end of subprogram
M911 thru M919	multiple block delete on
M921 thru M929	multiple block delete off
M950	vary tool offsets

* Startup Mode