

Table 6.7 List of G Codes

G code	Group	Function	B: Basic O: Optional	G code	Group	Function	B: Basic O: Optional
G00	01	Positioning	B	G54	12	Shift to work coordinate system 1	O
G01		Linear interpolation	B	G55		Shift to work coordinate system 2	O
G02		Circular interpolation, CW, Helical interpolation CW	B, O	G56		Shift to work coordinate system 3	O
G03		Circular interpolation CCW, Helical interpolation CCW	B, O	G57		Shift to work coordinate system 4	O
G04	02	Dwell	B	G58		Shift to work coordinate system 5	O
G06		Positioning in error detect off mode	B	G59		Shift to work coordinate system 6	O
G09		Exact stop	B	G60	01	Unidirectional approach	O
G10	*	Tool offset value and work coordinate, Shift-value modification	B, O	G61	13	Exact stop mode	B
G12	02	Circle cutting CW	O	G64		Exact stop mode cancel	B
G13		Circle cutting CCW	O	G65	*	Non-modal call of user macro	O
G17		XY plane designation	B	G66	14	Modal call of user macro	O
G18		ZX plane designation	B	G67		Modal call of user macro cancel	O
G19	06	YZ plane designation	B	G70	*	Bolt hole circle	O
G20		Inch input designation	O	G71		Arc	O
G21		Metric input designation	O	G72		Line-at-angle	O
G22	04	Stored stroke limit ON	O	G73	09	Canned cycle 10	O
G23		Stored stroke limit OFF	O	G74		Canned cycle 11	O
G27	09	Reference point check	O	G76		Canned cycle 12	O
G28		Automatic return to reference point	O	G77		Canned cycle 13	O
G29		Return from reference point	O	G80		Canned cycle cancel	O
G30		Return to 2nd, 3rd, 4th reference point	O	G81		Canned cycle 1, Output for external motion	O
G31	01	Skip function	O	G82		Canned cycle 2	O
G33		Thread cutting	O	G83		Canned cycle 3	O
G36		Automatic centering	O	G84		Canned cycle 4	O
G37		Automatic centering	O	G85	09	Canned cycle 5	O
G38	07	Z-axis reference surface offset	O	G86		Canned cycle 6	O
G40		Tool radius compensation cancel	O	G87		Canned cycle 7	O
G41		Tool radius compensation, left	O	G88		Canned cycle 8	O
G42	08	Tool radius compensation, right	O	G89		Canned cycle 9	O
G43		Tool length compensation, plus direction	B	G90	03	Absolute command designation	B
G44		Tool length compensation, minus direction	B	G91		Incremental command designation	B
G49		Tool length compensation, cancel	B	G92	*	Programming of absolute zero point	B
G45	08	Tool position offset, extension	B	G94	05	Feed per minute (mm/min) designation	O
G46		Tool position offset, retraction	B	G95		Feed per revolution (mm/rev.) designation	O
G47		Tool position offset, double extension	B	G98	10	Return to initial point for canned cycles	O
G48		Tool position offset, double retraction	B	G99		Return to point R for canned cycles	O
G50	15	Scaling OFF	O	G100	16	High-speed cutting cancel	O
G51		Scaling ON	O	G101		High-speed cutting in sequential processing mode ON	O
G52	12	Return to base coordinate system	O	G102		High-speed cutting in processing mode ON	O
G53	*	Temporary shift to machine coordinate system	O				

Notes:

1. The G codes in the * group are non-modal, and the effective only for the block in which they are commanded. They cannot be programmed twice or more in a block. They must be programmed only once in a block of its own.

2. The codes marked with ◼ is automatically selected at power on or reset.