

Siemens Sinumerik 808D G Codes (ISO dialect mode)



G Code List

G code	Description
G00	Rapid traverse
G01	Linear movement
G02	Circle/helix in clockwise direction
G03	Circle/helix in the counterclockwise direction
G04	Dwell time in [s] or spindle revolutions
G05	High-speed cycle cutting
G05.1	High-speed cycle -> Call CYCLE305
G08	Pre-control ON/OFFG15 Polar coordinates off
G09	Exact stop
G10	Write work offset/tool offset
G10.6	Retraction from contour (POLF)
G11	End parameter entry
G16	Polar coordinates on
G17	XY plane
G18	ZX plane

G19	YZ plane
G20	Inch input system
G21	Metric input system
G27	Checking the reference position
G28	1. Approaching a reference point
G30	2./3./4. Approaching a reference point
G30.1	Reference point position
G31	Measuring with "delete distance-to-go"
G40	Deselection of cutter radius compensation
G41	Compensation left of contour
G42	Compensation to right of contour
G43	Positive tool length compensation on
G44	Negative tool length compensation on
G49	Tool length compensation off
G50	Scaling off
G51	Scaling on
G50.1	Mirroring on programmed axis OFF
G51.1	Mirroring on programmed axis ON
G52	programmable work offset
G53	Approach position in machine coordinate system
G54 P0	external work offset
G54	Selecting work offset
G55	Selecting work offset
G56	Selecting work offset
G57	Selecting work offset
G58	Selecting work offset

G59	Selecting work offset
G60	directed positioning
G61	Exact stop modal
G63	Tapping mode
G64	Continuous-path modeG66 Macro module call
G65	Macro call
G67	Delete macro module call
G68	Rotation ON, 2D/3D
G69	Rotation OFF
G72.1	Contour repetition with rotation
G72.2	Linear contour repetition
G73	High-speed deep hole drilling cycle with chip breakage
G74	Left tapping cycle
G76	Fine drill cycle
G80	Cycle off
G81	Drilling cycle counterboring
G82	Countersink drilling cycle
G83	Deep hole drilling cycle with chip removal
G84	Right tapping cycle
G85	Boring cycle, retraction with G01 after reaching the end in axis Z, without spindle stop
G86	Boring cycle, spindle stops and then retraction with G00 after reaching the end in axis Z
G87	Reverse countersinking
G89	Boring cycle, stay for a while and then retraction with G01, without spindle rotation direction change
G90	Absolute programming
G91	Incremental programming
G92	Setting actual value

G92.1	Delete actual value, reset the WKS
G93	inverse-time feedrate (1/min)
G94	Feedrate in [mm/min, inch/min]
G95	Revolutional feedrate in [mm/rev, inch/rev]
G96	constant cutting rate on
G97	constant cutting rate off
G98	Return to starting point in fixed cycles
G99	Return to point R in fixed cycles
G290	Selection of Siemens mode
G291	Selection of ISO dialect mode

M Codes

M0	Operation Stop
M1	Program Stop
M2	Program End
M3	Spindle CW
M4	Spindle CCW
M5	Spindle Stop
M6	Load Tool
M8	Coolant On
M9	Coolant off
M17	Sub Program Return
M30	Program End
M87	Air Blast
M98	Sub Program Call
M99	Sub Program Return

*M Codes vary from machine to machine depending on options.