

## G-Codes - Milling

G00	Linear Rapid Positioning
G01	Linear Feed Interpolation
G02	CW Circular/Helical Interpolation
G03	CCW Circular/Helical Interpolation
G04	Dwell
G05	High Speed Cycle Machining
G06	NURBS Machining
G07	Imaginary Axis Designation
G08	Fast Cornering Mode
G09	Exact Stop
G10	Offset Value Setting
G11	Offset Value Setting Cancel
G15	Polar coordinates command cancel
G16	Polar coordinates command
G17	XY Plane Selection
G18	ZX Plane Selection
G19	YZ Plane Selection
G20	Input in inch
G21	Input in mm
G22	Stored Stroke Limit On
G23	Stored Stroke Limit Off
G24	Circular Pocket Clear
G25	Circular Finish Inside
G26	Circular Finish Outside
G27	Reference Point Return Check
G28	Return to Reference Point
G29	Return from Reference Point
G30	Return to 2nd, 3rd & 4th Reference Point
G31	Skip Function
G32	Z to Tool Change
G33	Thread Cutting
G34	Rectangular Pocket Clear
G35	Rectangular Finish Inside
G36	Rectangular Finish Outside
G39	Corner Offset Circular Interpolation
G40	Cutter Compensation Cancel
G41	Cutter Compensation Left
G42	Cutter Compensation Right
G43	Tool Length Compensation + Direction
G44	Tool Length Compensation - Direction
G45	Tool Offset Increase
G46	Tool Offset Decrease
G47	Tool Offset Double Increase
G48	Tool Offset Double Decrease
G49	Tool Length Compensation Cancel

G50	Scaling Off
G51	Scaling On
G52	Local Coordinate System Setting
G53	Machine Coordinate System Selection
G54	Work Coordinate System 1 Selection
G55	Work Coordinate System 2 Selection
G56	Work Coordinate System 3 Selection
G57	Work Coordinate System 4 Selection
G58	Work Coordinate System 5 Selection
G59	Work Coordinate System 6 Selection
G60	Single Direction Positioning
G61	Exact Stop Mode
G62	Automatic Corner Override
G63	Tapping Mode
G64	Cutting Mode
G65	Simple Macro Call
G66	Custom Macro Modal Call
G67	Custom Macro Modal Call Cancel
G68	Coordinate System Rotation
G69	Coordinate System Rotation Cancel
G70	Input in inch
G71	Input in mm
G73	Peck Drilling Cycle
G74	Counter Tapping Cycle
G76	Fine Boring
G80	Canned Cycle Cancel
G81	Drilling Cycle, Spot Boring
G82	Drilling Cycle, Counter Boring
G83	Peck Drilling Cycle
G84	Tapping Cycle
G85	Boring Cycle - Feed Out
G86	Boring Cycle - Stop, Rapid Out
G87	Back Boring Cycle
G88	Boring Cycle
G89	Boring Cycle - Dwell, Feed Out
G90	Absolute Programming
G91	Incremental Programming
G92	Programming Of Absolute Zero Point
G94	Feed Per Minute
G95	Feed Per Revolution
G96	Constant Surface Speed Control
G97	Constant Surface Speed Control Cancel
G98	Return To Initial Point In Canned Cycle
G99	Return To Ref Point In Canned Cycle

## G-Codes - Turning

G00	Linear Rapid Positioning
G01	Linear Feed Interpolation
G02	CW Circular/Helical Interpolation
G03	CCW Circular/Helical Interpolation
G04	Dwell
G07	Imaginary Axis Interpolation, Sine
G09	Curve
G10	Exact Stop
G11	Offset Value Setting Offset Value Setting Cancel
G20	Input in inch
G21	Input in mm
G22	Stored Stroke Limit On
G23	Stored Stroke Limit Off
G27	Reference Point Return Check
G28	Return to Reference Point
G29	Return from Reference Point
G30	Return to 2nd, 3rd & 4th Reference Point
G31	Skip Function
G32	Thread Cutting
G34	Variable Lead Thread Cutting
G36	Automatic Tool Compensation X
G37	Automatic Tool Compensation Z

G40	Tool Nose Radius Compensation Cancel
G41	Tool Nose Radius Compensation Left
G42	Tool Nose Radius Compensation Right
G50	Programming Of Absolute Zero
G65	Simple Macro Call
G66	Custom Macro Modal Call
G67	Custom Macro Modal Call Cancel
G68	Mirror Image For Double Turrets On
G69	Mirror Image For Double Turrets Off
G70	Finishing Canned Cycle
G71	Rough Turning Canned Cycle
G72	Face Turning Canned Cycle
G73	Pattern Repeating
G74	Peck Drilling in Z-Axis
G75	Grooving in X-Axis
G76	Thread Cutting Cycle
G90	Cutting Cycle A
G92	Thread Cutting Cycle
G94	Cutting Cycle B
G96	Constant Surface Speed Control
G97	Constant Surface Speed Control Cancel
G98	Feed Per Minute
G99	Feed Per Revolution