

G-Code Quick Reference -Mill

G0 - Rapid Movement
G1 - Linear Movement
G2 - Interpolation Clockwise
G3 - Interpolation Counter Clockwise

G4 - Dwell
G9 - Exact Stop/Exact Position
G10 - Data Setting

G17 - XY plane selection
G18 - ZX plane selection
G19 - YZ plane selection

G20 - Machine in inch
G21 - Machine in MM

G28 - Return to Reference Position
G30 - Return to 2nd reference position

G40 - Cutter Compensation Cancel
G41 - Cutter Compensation Left
G42 - Cutter Compensation Right

G43 - Tool Length Compensation +
G44 - Tool Length Compensation -
G49 - Tool Length Cancel

G50 - Scaling Cancel
G51 - Scaling

G54-G59 - Work Coordinate Systems

G68 - Coordinate Rotation
G69 - Coordinate Rotation Cancel

G90 - Absolute Command
G91 - Incremental Command
G92 - Programming of Absolute Zero

G94 - Feed Per Minute
G95 - Feed Per Revolution

G98 - Return to initial point in canned cycle
G99 - Return to R point in canned cycle

G80 - Cancel Canned Cycle
G73 - Peck Drilling Cycle
G74 - Left Hand Tapping Cycle
G76 - Boring Cycle
G81 - Drilling Cycle
G82 - Drilling Cycle
G83 - Pecking Cycle
G84 - Right Hand Tapping Cycle
G85 - Boring Cycle
G86 - Boring Cycle
G87 - Back Boring Cycle