

# CNC G codes

G00 - Positioning at rapid speed; Mill and Lathe

G01 - Linear interpolation (machining a straight line); Mill and Lathe

G02 - Circular interpolation clockwise (machining arcs); Mill and Lathe

G03 - Circular interpolation, counter clockwise; Mill and Lathe

G04 - Mill and Lathe, Dwell

G09 - Mill and Lathe, Exact stop

G10 - Setting offsets in the program; Mill and Lathe

G12 - Circular pocket milling, clockwise; Mill

G13 - Circular pocket milling, counterclockwise; Mill

G17 - X-Y plane for arc machining; Mill and Lathe with live tooling

G18 - Z-X plane for arc machining; Mill and Lathe with live tooling

G19 - Z-Y plane for arc machining; Mill and Lathe with live tooling

G20 - Inch units; Mill and Lathe

G21 - Metric units; Mill and Lathe

G27 - Reference return check; Mill and Lathe

G28 - Automatic return through reference point; Mill and Lathe

G29 - Move to location through reference point; Mill and Lathe (slightly different for each machine)

G31 - Skip function; Mill and Lathe

G32 - Thread cutting; Lathe

G33 - Thread cutting; Mill

G40 - Cancel diameter offset; Mill. Cancel tool nose offset; Lathe

G41 - Cutter compensation left; Mill. Tool nose radius compensation left; Lathe

G42 - Cutter compensation right; Mill. Tool nose radius compensation right; Lathe

G43 - Tool length compensation; Mill

G44 - Tool length compensation cancel; Mill (sometimes G49)

G50 - Set coordinate system and maximum RPM; Lathe

G52 - Local coordinate system setting; Mill and Lathe

G53 - Machine coordinate system setting; Mill and Lathe

G54~G59 - Workpiece coordinate system settings #1 to #6; Mill and Lathe

G61 - Exact stop check; Mill and Lathe

G65 - Custom macro call; Mill and Lathe

G70 - Finish cycle; Lathe

G71 - Rough turning cycle; Lathe

G72 - Rough facing cycle; Lathe

G73 - Irregular rough turning cycle; Lathe

G73 - Chip break drilling cycle; Mill

G74 - Left hand tapping; Mill

G74 - Face grooving or chip break drilling; Lathe

G75 - OD groove pecking; Lathe

G76 - Fine boring cycle; Mill

G76 - Threading cycle; Lathe

G80 - Cancel cycles; Mill and Lathe

G81 - Drill cycle; Mill and Lathe

G82 - Drill cycle with dwell; Mill

G83 - Peck drilling cycle; Mill

G84 - Tapping cycle; Mill and Lathe

G85 - Bore in, bore out; Mill and Lathe

G86 - Bore in, rapid out; Mill and Lathe

G87 - Back boring cycle; Mill

G90 - Absolute programming

G91 - Incremental programming

G92 - Reposition origin point; Mill

G92 - Thread cutting cycle; Lathe

G94 - Per minute feed; Mill

G95 - Per revolution feed; Mill

G96 - Constant surface speed control; Lathe

G97 - Constant surface speed cancel

G98 - Per minute feed; Lathe

G99 - Per revolution feed; Lathe

# CNC M Codes

- M00 - Program stop; Mill and Lathe
- M01 - Optional program stop; Lathe and Mill
- M02 - Program end; Lathe and Mill
- M03 - Spindle on clockwise; Lathe and Mill
- M04 - Spindle on counterclockwise; Lathe and Mill
- M05 - Spindle off; Lathe and Mill
- M06 - Tool change; Mill
- M08 - Coolant on; Lathe and Mill
- M09 - Coolant off; Lathe and Mill
- M10 - Chuck or rotary table clamp; Lathe and Mill
- M11 - Chuck or rotary table clamp off; Lathe and Mill
- M19 - Orient spindle; Lathe and Mill
- M30 - Program end, return to start; Lathe and Mill
- M97 - Local sub-routine call; Lathe and Mill
- M98 - Sub-program call; Lathe and Mill
- M99 - End of sub program; Lathe and Mill