

G AND M CODE CHEAT SHEET

G00 - Positioning at rapid speed; Mill and Lathe	G49 - Tool length compensation cancel; Mill
G01 - Linear interpolation (machining a straight line); Mill and Lathe	G50 - Set coordinate system and maximum RPM; Lathe
G02 - Circular interpolation clockwise (machining arcs); Mill and Lathe	G52 - Local coordinate system setting; Mill and Lathe
G03 - Circular interpolation, counter clockwise; Mill and Lathe	G53 - Machine coordinate system setting; Mill and Lathe
G04 - Mill and Lathe, Dwell	G54~G59 - Workpiece coordinate system settings #1 to #6; Mill and Lathe
G09 - Mill and Lathe, Exact stop	G61 - Exact stop check; Mill and Lathe
G10 - Setting offsets in the program; Mill and Lathe	G65 - Custom macro call; Mill and Lathe
G12 - Circular pocket milling, clockwise; Mill	G70 - Finish cycle; Lathe
G13 - Circular pocket milling, counterclockwise; Mill	G71 - Rough turning cycle; Lathe
G17 - X-Y plane for arc machining; Mill and Lathe with live tooling	G72 - Rough facing cycle; Lathe
G18 - Z-X plane for arc machining; Mill and Lathe with live tooling	G73 - Irregular rough turning cycle; Lathe
G19 - Z-Y plane for arc machining; Mill and Lathe with live tooling	G73 - Chip break drilling cycle; Mill
G20 - Inch units; Mill and Lathe	G74 - Left hand tapping; Mill
G21 - Metric units; Mill and Lathe	G74 - Face grooving or chip break drilling; Lathe
G27 - Reference return check; Mill and Lathe	G75 - OD groove pecking; Lathe
G28 - Automatic return through reference point; Mill and Lathe	G76 - Fine boring cycle; Mill
G29 - Move to location through reference point; Mill and Lathe (slightly different for each machine)	G76 - Threading cycle; Lathe
G31 - Skip function; Mill and Lathe	G80 - Cancel cycles; Mill and Lathe
G32 - Thread cutting; Lathe	G81 - Drill cycle; Mill and Lathe
G33 - Thread cutting; Mill	G82 - Drill cycle with dwell; Mill
G40 - Cancel diameter offset; Mill. Cancel tool nose offset; Lathe	G83 - Peck drilling cycle; Mill
G41 - Cutter compensation left; Mill. Tool nose radius compensation left; Lathe	G84 - Tapping cycle; Mill and Lathe
G42 - Cutter compensation right; Mill. Tool nose radius compensation right; Lathe	G85 - Bore in, bore out; Mill and Lathe
G43 - Tool length compensation; Mill	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 - Initial level return in canned cycles; Mill
	G99 - R value return in canned cycles; Mill
M00 - Program stop; Mill and Lathe	M09 - Coolant off; Lathe and Mill
M01 - Optional program stop; Lathe and Mill	M10 - Chuck or rotary table clamp; Lathe and Mill
M02 - Program end; Lathe and Mill	M11 - Chuck or rotary table clamp off; Lathe and Mill
M03 - Spindle on clockwise; Lathe and Mill	M19 - Orient spindle; Lathe and Mill
M04 - Spindle on counterclockwise; Lathe and Mill	M30 - Program end, return to start; Lathe and Mill
M05 - Spindle off; Lathe and Mill	M97 - Local sub-routine call; Lathe and Mill
M06 - Toolchange; Mill	M98 - Sub-program call; Lathe and Mill
M08 - Coolant on; Lathe and Mill	M99 - End of sub program; Lathe and Mill