

Generic G Code List

G00 - Rapid Positioning	G52 - Local Coordinate Setting
G01 - Linear Interpolation	G53 - Machine Coordinate Setting
G02 - Circular Interpolation CW	G54 - Workpiece Coordinate Setting 1
G03 - Circular Interpolation CCW	G55 - Workpiece Coordinate Setting 2
G04 - Dwell	G56 - Workpiece Coordinate Setting 3
G05 - High Speed Machining	G57 - Workpiece Coordinate Setting 4
G07 - Imaginary Axis Designation	G58 - Workpiece Coordinate Setting 5
G09 - Exact Stop Check	G59 - Workpiece Coordinate Setting 6
G10 - Program Parameter Input	G60 - Single Direction Positioning
G11 - Program Parameter Input Cancel	G62 - Automatic Corner Override
G12 - Circle Cutting CW	G63 - Tapping Mode
G13 - Circle Cutting CCW	G64 - Cutting Mode
G17 - XY Plane Selection	G65 - User Simple Macro Call
G18 - XZ Plane Selection	G66 - User Modal Macro Call
G19 - YZ Plane Selection	G67 - User Modal Macro Call Cancel
G20 - Input in Inches	G68 - Coordinate System Rotation On
G21 - Input in Metric	G69 - Coordinate System Rotation Off
G22 - Stored Stroke Limit ON	G70 - Input in Inch
G23 - stored Stroke Limit OFF	G71 - Input in Metric
G27 - Reference Point Return Check	G73 - High Speed Peck Drilling Cycle
G28 - Automatic Return to Reference Point	G74 - Left Hand Tapping Cycle
G29 - Automatic Return from Reference Point	G76 - Fine Boring Cycle
G30 - Return to 2nd, 3rd, 4th Reference Point	G80 - Canned Cycle Cancel
G31 - Skip Function	G81 - Drilling Cycle
G33 - Thread Cutting	G82 - Counter Boring Cycle
G34 - Bolt Hole Circle (Canned Cycle)	G83 - Peck Drilling Cycle
G35 - Line at Angle (Canned Cycle)	G84 - Right Hand Tapping Cycle
G36 - Arc (Canned Cycle)	G85 - Boring Cycle
G40 - Cutter Compensation Cancel	G87 - Back Boring Cycle
G41 - Cutter Compensation Left	G90 - Absolute Positioning
G42 - Cutter Compensation Right	G91 - Incremental Positioning
G43 - Tool Length Compensation +	G92 - Reposition Origin Point
G44 - Tool Length Compensation -	G94 - Feed Per Minute
G45 - Tool Offset Increase	G95 - Feed Per Revolution
G46 - Tool Offset Decrease	G96 - Constant Surface Speed Control On
G49 - Tool Length Compensation Cancel	G97 - Constant Surface Speed Control Off
G50 - Scaling Off	G98 - Set Initial Plane
G51 - Scaling On	G99 - Return to R Plane