**CNC General Codes**

**LAB 5**

**Haas Lathe G Codes List - Haas G Codes for CNC Lathes**

**G00: Rapid Position Motion**

**G01: Linear Interpolation Motion or Linear Motion, Chamfer and Corner Rounding – Modal**

**G02: CW Circulation Interpolation Motion – Modal**

**G03: CCW Circular Interpolation Motion – Modal**

**G04: Dwell (P) P=Seconds. Milliseconds**

**G05: Fine Spindle Control Motion (Live Tooling) – Optional**

**G09: Exact Stop**

**G10: Programmable Offset Setting**

**G14: Main-Spindal / Sub-Spindle Swap – Optional**

**G15: Main-Spindal / Sub-Spindle Swap Cancel – Optional**

**G17: Circular Motion XYZ Plane Selection Live Tooling**

**(G02, G03) – Modal, Optional**

**G18: Circular Motion ZX Plane Selection (G02, G03)**

**G19: Circular Motion YZ Plane Selection Live Tooling (G02, G03) – Modal/Optional**

**G20: Verify Inch Coordinate Positioning**

**G21: Verify Metric Coordinate Positioning – Modal**

**G28: Rapid to machine zero return through Ref. Point**

**G29: Move to Location Through G29 Ref. Point**

**G31: Feed Until skip Function – Optional**

**G32: Thread cutting path – Modal**

**G40: Tool Nose Compensation Cancel G41/G42**

**G41: Tool Nose Compensation, Left – Modal**

**G42: Tool Nose Compensation, Right – Modal**

**G50: Spindle Speed Maximum RPM Limit (S) – Modal**

**G51: Rapid to Machine zero, Cancel offset**

**G52: Work offset Positioning Coordinate OR Global Work Offset Coordinate System Shift – Modal**

**G53: Machine Zero Positioning Coordinate**

**G54: Work Offset Positioning Coordinate #1**

**G55: Work Offset Positioning Coordinate #2 – Modal**

**G56: Work Offset Positioning Coordinate #3 – Modal**

**G57: Work Offset Positioning Coordinate #4 – Modal**

**G58: Work Offset Positioning Coordinate #5 – Modal**

**G59: Work Offset Positioning Coordinate #6 – Modal**

**G61: Exact Stop – Modal**

**G64: Exact Stop G61 Cancel**

**G65: Macro Sub-Routine Call – Optional**

**G70: Finishing Cycle**

**G71: O.D. / I.D. Stock Removal Cycle Example**

**G72: End Face Stock Removal Cycle**

**G73: Irregular Path Stock Removal Cycle**

**G74: Face Grooving OR High Speed Peck Drill Cycle**

**G75: Peck Grooving Cycle O.D. or I.D.**

**G76: Threading Cycle, Multiple Pass O.D. / I.D**

**G77: Live Tooling Flatting Cycle – Optional**

**G80: Cancel Canned Cycle**

**G81: Drill Canned Cycle – Modal**

**G82: Spot Drill / Counterbore Canned Cycle – Modal**

**G83: Peck Drill Deep Hole Canned Cycle – Modal**

**G84: Tapping Canned Cycle – Modal**

**G85: Bore In-Bore Out Canned Cycle – Modal**

**G86: Bore In-Stop-Rapid Out Canned Cycle – Modal**

**G87: Bore In-Stop-Manual Retract Canned Cycle – Modal**

**G88: Bore In-Dwell-Manual Retract Canned Cycle – Modal**

**G89: Bore In-Dwell-Bore-out Canned Cycle – Modal**

**G90: O.D. / I.D. Turning Cycle – Modal**

**G92: Threading Cycle – Modal**

**G94: End Facing Cycle – Modal**

**G95: Live Tooling End Face Rigid Tap – Modal/Optional**

**G96: Constant Surface speed, CSS On – Modal**

**G97: Constant Non-Varying Spindle Speed, CSS Off (S)**

**G98: Feed Per Minute (F) – Modal**

**G99: Feed Per Revolution (F)**

**G100: Mirror Image Cancel G101**

**G101: Mirror Image**

**G102: Programmable Output to RS – 232**

**G103: Limit Block Lookahead**

**G105: Servo Bar Command – Optional**

**G110-G111: Work Offset-positioning Coordinate #7-#8 – Modal**

**G112: Cartesian to Polar Transformation – Optional**

**G113: Cartesian to Polar Transformation Cancel – Optional**

**G114-G129: Work Offset Positioning Coordinate #9-#24 – Modal**

**G154: Select Work Offset Positioning Coordinate P1-99 (P) – Modal**

**G159: Background Pickup / Part Return – Optional**

**G160: APL Axis Command On – Optional**

**G161: APL Axis Command Off – Optional**

**G184: Reverse Tapping Canned Cycle – Modal**

**G186: Live Tooling Reverse Rigid Tap – Optional**

**G187: Accuracy Control for High Speed Machining (E)**

**G194: Sub-Spindle / Tapping Canned Cycle – Modal**

**G195: Live Tooling Radial Tapping – Optional**

**G196: Live Tooling Radial Tapping Reverse – Optional**

**G200: Index on the Fly**

**Haas Lathe M Codes List - Haas M Codes for CNC Lathes**

**M00: Program Stop – Modal**

**M01: Optional Program Stop – Modal**

**M02: Program End – Modal**

**M03: Spindle on Forward (S) – Modal**

**M04: Spindle on Reverse (S) – Modal**

**M05: Spindle Stop – Modal**

**M08: Coolant On – Modal**

**M09: Coolant Off – Modal**

**M10: Chuck Clamp – Modal**

**M11: Chuck Unclamp – Modal**

**M12: Auto Air Jet On (P) – Modal/Optional**

**M13: Auto Air Jet Off – Modal/Optional**

**M14: Main Spindle Clamp – Modal/Optional**

**M15: Main Spindle Unclamp – Modal/Optional**

**M17: Rotate Turret Forward (T) – Modal**

**M18: Rotate turret Reverse (T) – Modal**

**M19: Orient Spindle – Modal/Optional Example**

**M21: Tailstock Advance – Modal/Optional**

**M22: Trailstock Retract – Modal/Optional**

**M23: Angle Out of Thread On – Modal**

**M24: Angle of Thread Off – Modal**

**M25-M28: Optional User M Code Interface with M-Fin Signal – Modal**

**M30: Program End and Reset – Modal**

**M31: Chip Auger Forward – Modal**

**M33: Chip Auger Stop – Modal**

**M36: Parts Catcher On – Modal/Optional**

**M37: Parts Catcher Off – Modal/Optional**

**M38: Specify Spindle Variation On – Modal**

**M39: Specify Spindle Variation Off – Modal**

**M41: Spindle Low Gear Override – Modal**

**M42: Spindle High Gear Override – Modal**

**M43: Turret Unlock – Modal**

**M44: Turret Lock – Modal**

**M51-M58: Optional User M Code Set – Modal**

**M59: Output Relay Set (N) – Modal**

**M61-M68: Optional User M Code Clear – Modal**

**M69: Output Relay Clear (N) – Modal**

**M76: Program Displays Inactive – Modal**

**M77: Program Displays Active – Modal**

**M78: Alarm in Skip Signal Found – Modal**

**M79: Alarm if Skip signal Not Found – Modal**

**M85: Automatic Door Open – Modal/Optional**

**M86: Automatic Door Close – Modal/Optional**

**M88: High Pressure Coolant ON – Modal/Optional**

**M89: High Pressure Coolant off – Modal/Optional**

**M93: Axis Position Capture Start – Modal/Optional**

**M94: Axis Position Capture Stop – Modal/Optional**

**M95: Sleep Mode – Modal**

**M96: Jump if no Signal – Modal**

**M97: Local Sub-Routine Call – Modal**

**M98: Sub-Program Call – Modal**

**M99: Sub-Program / Routine Return or Loop – Modal**

**M109: Inactive User Input (P) – Modal/Optional**

**M110: Tailstock Chuck Clamp – Modal/Optional**

**M111: Tailstock Chuck Unclamp – Modal/Optional**

**M119: Sub-Spindle Orient – Modal/Optional\*\***

**M121-M128: Optional User M Code Interface with M-Fin Signal – Modal**

**M133: Live tool Drive Forward (P) – Modal/Optional Example**

**M134: Live Tool Drive Reverse (P) – Modal/Optional**

**M135: Live Tool Drive Stop – Modal/Optional**

**M143: Sub-Spindle Forward (P) – Modal/Optional**

**M144: Sub-Spindle Reverse (P) – Modal/Optional**

**M145: Sub-Spindle Stop – Modal/Optional**

**M154: C Axis Engage – Modal/Optional Example**

**M155: C Axis Disengage – Modal/Optional**

**M164: Rotate APL Grippers To “n” Position – Modal/Optional**

**M165: Open APL Gripper 1 (Raw Material) – Modal/Optional**

**M166: Close APL Gripper 1 (Raw Material) – Modal/Optional**

**M167: Open APL Gripper 2 Finished Material) – Modal/Optional**

**M168: Close APL Gripper 2 (Finished Material) – Modal/Optional**

**Haas Mill G Codes List - Haas G Codes for CNC Milling**

**G00: Rapid Motion**

**G01: Linear Interpolation Motion**

**G02: CW Interpolation Motion**

**G03: CCW Interpolation Motion**

**G04: Dwell**

**G09: Exact Stop**

**G10: Programmable Offset Setting**

**G12: CW Circular Pock Milling (Yasnac)**

**G13: CCW Circular Pock Milling (Yasnac)**

**G17: XY Plane Selection**

**G18: ZX Plane Selection**

**G19: YZ Plane Selection**

**G20: Inch Programming Selection**

**G21: Metric Programming Selection**

**G28: Return to Machine Zero**

**G29: Move to Location Through G29 Reference**

**G31: Skip Function**

**G35: Automatic Tool Diameter Measurement**

**G36: Automatic Work Offset Measurement**

**G37: Automatic Tool Length Measurement**

**G40: Cutter Comp Cancel**

**G41: Cutter Compensation Left**

**G42: Cutter Compensation Right**

**G43: Tool Length Compensation +**

**G44: Tool Length Compensation –**

**G47: Engraving**

**G49: G43/G44 Cancel**

**G50: G51 Cancel**

**G51: Scaling**

**G52: Select Work Coordinate System G52 (Yasnac)**

**G52: Set Local Coordinate System (Fanuc)**

**G52: Set Local Coordinate System (HAAS)**

**G53: Non-Modal Machine Coordinate Selection**

**G54: Select Work Coordinate System l**

**G55: Select Work Coordinate System 2**

**G56: Select Work Coordinate System 3**

**G57: Select Work Coordinate System 4**

**G58: Select Work Coordinate System 5**

**G59: Select Work Coordinate System 6**

**G60: Unidirectional Positioning**

**G61: Exact Stop Modal**

**G64: G61 Cancel**

**G65: Macro Subroutine Call**

**G68: Rotation**

**G69: G68 Cancel**

**G70: Bolt Hole Circle (Yasnac) Example1 Example2 Example3**

**G71: Bolt Hole Arc (Yasnac) Example**

**G72: Bolt Holes Along an Angle (Yasnac) Example**

**G73: High Speed Peck Drill Canned Cycle**

**G74: Reverse Tap Canned Cycle**

**G76: Fine Boring Canned Cycle**

**G77: Back Bore Canned Cycle**

**G80: Canned Cycle Cancel**

**G81: Drill Canned Cycle Example1 Example2 Ex3 Ex4 Ex5**

**G82: Spot Drill Canned Cycle Ex1**

**G83: Peck Drill Canned Cycle Ex1 Ex2**

**G84: Tapping Canned Cycle Example1 Peck Tapping**

**G85: Boring Canned Cycle**

**G86: Bore/Stop Canned Cycle**

**G87: Bore/Manual Retract Canned Cycle**

**G88: Bore/Dwell Canned Cycle**

**G89: Bore Canned Cycle**

**G90: Absolute**

**G91: Incremental**

**G92: Set Work Coordinates – FANUC or HAAS**

**G92: Set Work Coordinates – YASNAC**

**G93: Inverse Time Feed Mode ON**

**G94: Inverse Time Feed Mode OFF/Feed Per Minute ON**

**G98: Initial Point Return**

**G99: R Plane Return**

**G100: Disable Mirror Image**

**G101: Enable Mirror Image**

**G102: Programmable Output To RS-232**

**G103: Block Look ahead Limit**

**G107: Cylindrical Mapping**

**G110: Select Coordinate System 7**

**G111: Select Coordinate System 8**

**G112: Select Coordinate System 9**

**G113: Select Coordinate System 10**

**G114: Select Coordinate System 11**

**G115: Select Coordinate System 12**

**G116: Select Coordinate System 13**

**G117: Select Coordinate System 14**

**G118: Select Coordinate System 15**

**G119: Select Coordinate System 16**

**G120: Select Coordinate System 17**

**G121: Select Coordinate System 18**

**G122: Select Coordinate System 19**

**G123: Select Coordinate System 20**

**G124: Select Coordinate System 21**

**G125: Select Coordinate System 22**

**G126: Select Coordinate System 23**

**G127: Select Coordinate System 24**

**G128: Select Coordinate System 25**

**G129: Select Coordinate System 26**

**G136: Automatic Work Offset Center Measurement**

**G141: 3D+ Cutter Compensation**

**G143: 5 Axis Tool Length Compensation+**

**G150: General Purpose Pocket Milling**

**G153: 5 Axis High Speed Peck Drill Canned Cycle**

**G154: P1-P99 Replaces G110-G129 on newer machines**

**G155: 5 Axis Reverse Tapping Canned Cycle**

**G161: 5 Axis Drill Canned Cycle**

**G162: 5 Axis Spot Drill/Counterbore Canned Cycle**

**G163: 5 Axis Peck Drill Canned Cycle (Setting 22)**

**G164: 5 Axis Tapping Canned Cycle**

**G165: 5 Axis Bore in, Bore out Canned Cycl**

**G166: 5 Axis Bore in, Stop, Rapid out Canned Cycle**

**G169: 5 Axis Bore, Dwell, Bore out Canned Cycle**

**G174: Special Purpose Non-Vertical Rigid Tapping CCW**

**G184: Special Purpose Non-Vertical Rigid Tapping CW**

**G187: Accuracy Control for High Speed Machining**

**G188: Get Program From PST (Program Schedule Table)**

**Haas Mill M Codes List - Haas M Codes for CNC Milling**

**M00: Program Stop**

**M01: Optional Program Stop**

**M02: Program End (Setting 39)**

**M03: Spindle On, Clockwise (S) (Setting 144)**

**M04: Spindle On, Counterclockwise (S) (Setting 144)**

**M05: Spindle Stop**

**M06: Tool Change (T) (Setting 42, 87, 155)**

**M08: Coolant On (Setting 32)**

**M09: Coolant Off**

**M10: 4th Axis Brake On**

**M11: 4th Axis Brake Release**

**M12: 5th Axis Brake On**

**M13: 5th Axis Brake Release**

**M16: Tool Change (T) (Same as M06)**

**M17: APC Pallet Unclamp and Open APC Door**

**M18: APC Pallet Clamp and Close APC Door**

**M19: Orient Spindle (P, R values optional)**

**M21-M28: Optional User M Code Interface with M-Fin Signals**

**M30: Program End and Reset (Setting 2, 39, 56, 83)**

**M31: Chip Auger Forward (Setting 114,115)**

**M33: Chip Auger Stop**

**M34: Coolant Spigot Position Down, Increment (+1)**

**M35: Coolant Spigot Position Up, Decrement (-1)**

**M36: Pallet Part Ready (P)**

**M39: Rotate Tool Turret (T#) (Setting 86)**

**M41: Spindle Low Gear Override**

**M42: Spindle High Gear Override**

**M50: Execute Pallet Change (P) (Setting 121 thru,129)**

**M51-M58: Optional User M Code Set**

**M59: Output Relay Set (N)**

**M61-M68: Optional User M Code Clear**

**M69: Output Relay Clear (N)**

**M75: Set G35 or G136 Reference Point**

**M76: Control Display Inactive**

**M77: Control Display Active**

**M78: Alarm if Skip Signal Found**

**M79: Alarm if Skip Signal Not Found**

**M80: Automatic Door Open (Setting 131)**

**M81: Automatic Door Close (Setting 131)**

**M82: Tool Unclamp**

**M83: Auto Air Jet On**

**M84: Auto Air Jet Off**

**M86: Tool Clamp**

**M88: Coolant Through the Spindle On**

**M89: Coolant Through the Spindle Off**

**M93: Axis POS Capture Start (P, Q)**

**M94: Axis POS Capture Stop**

**M95: Sleep Mode**

**M96: Jump if No Input (P, Q)**

**M97: Local Sub-Program Call (P, L)**

**M98: Sub Program Call (P, L)**

**M99: M97 Local Sub-Program or M98 Sub-Program Return or Loop Program (Setting 118)**

**M101: MOM (Minimum Oil Machining) CANNED CYCLE MODE (I)**

**M102: MOM (Minimum Oil Machining) MODE (I, J)**

**M103: MOM (Minimum Oil Machining) MODE CANEL**

**M109: Interactive User Input (P)**