CNC General Codes

LAB 12

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)