CNC PROGRAMMING WORKBOOK

CODE	FUNCTION
G00	Rapid traverse motion; Used for non-cutting rapid moves of the machine axis to a location to be machined, or rapid retract moves after cuts have been completed. Maximum rapid motion (I.P.M.) of a CNC Machine will vary on machine model.
G01	Linear interpolation motion; Used for actual machining and metal removal. Governed by a programmed feedrate in inches (or mm) per minute. Maximum feed rate (I.P.M.) of a CNC Machine will vary depending on the model of the machine.
G02	Circular Interpolation, Clockwise
G03	Circular Interpolation, Counterclockwise
G04	Dwell- Used with an X value for time of dwell in seconds
G18	ZX Plane Selection
G20	Verify Inch Coordinate Positions
G21	Verify Metric Coordinate Positions
G28	Machine Home (Rapid traverse)
G40	Tool Nose Radius Compensation CANCEL
G41	Tool Nose Radius Compensation LEFT of the programmed path
G42	Tool Nose Radius Compensation RIGHT of the programmed path
G50	Max RPM Preset
G52	Local Coordinate system setting
G53	Machine Zero Positioning Coordinate Shift
G54-59	Select Coordinate System #1 - #6

LESSON-2

INTRODUCTION TO CNC CODES

camInstructor

LESSON-2 – INTRODUCTION TO CNC CODES
LATHE TOOLS

The CNC Lathe used in this text is set-up with following tools. All program examples and exercises in this workbook are typically using the same tools.

Turret #	Tool Description
1	O.D. Right Hand Roughing Tool 80°
2	O.D. Right Hand Finishing Tool 55°
3	O.D. Profiling Tool 35°
4	Right Hand Parting Tool Width 0.118
5	O.D. Thread Tool
6	# 4 Centre Drill
7	1/4” Diameter Drill
8	Boring Tool

COMMONLY USED PREPARATORY **G** CODES

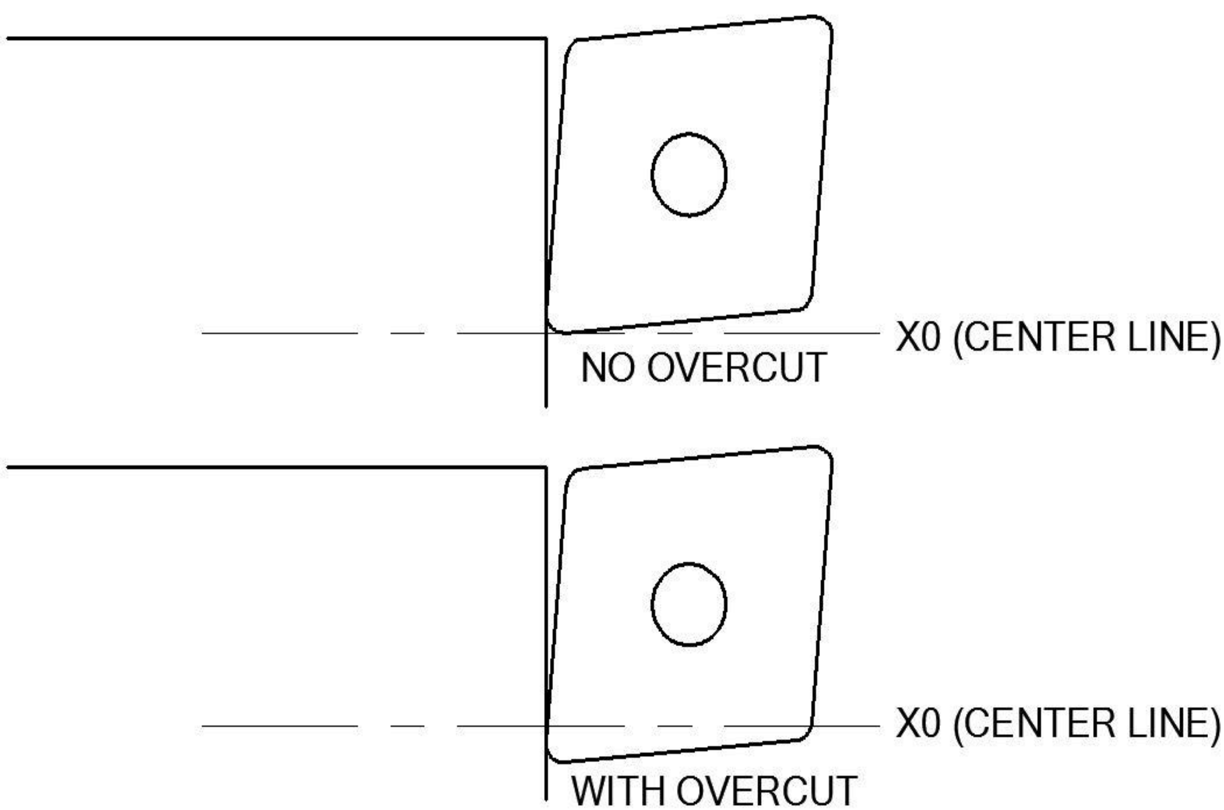
CODE	FUNCTION
G00	Rapid traverse motion; Used for non-cutting rapid moves of the machine axis to a location to be machined, or rapid retract moves after cuts have been completed.
G01	Linear interpolation motion; Used for actual machining and metal removal. Governed by a programmed feedrate in inches (or mm) per minute.
G02	Circular Interpolation, Clockwise
G03	Circular Interpolation, Counterclockwise
G04	Dwell- Used with an X value for time of dwell in seconds
G18	ZX Plane Selection
G20	Verify Inch Coordinate Positions
G21	Verify Metric Coordinate Positions
G28	Machine Home (Rapid traverse)
G40	Tool Nose Radius Compensation CANCEL
G41	Tool Nose Radius Compensation LEFT of the programmed path
G42	Tool Nose Radius Compensation RIGHT of the programmed path
G50	Max RPM Preset
G52	Local Coordinate system setting
G53	Machine Zero Positioning Coordinate Shift
G54-G59	Select Coordinate System #1 - #6 (Part zero offset location)
G70	Profile Finish Turning fixed cycle
G71	Profile Rough Turning fixed cycle – Z axis direction
G72	Profile Rough Turning fixed cycle – X axis direction
G73	Pattern Repetition cycle
G74	Drilling Cycle
G75	Grooving cycle
G76	Threading cycle
G80	Cancel Canned Cycle
G81	Drill Canned Cycle
G96	Constant Surface Speed (CSS)
G97	Direct RPM Input Mode (cancels CSS mode)
G98	Feed Rate per Minute
G99	Feed Rate per Revolution

PROGRAMMING NOTE

As you may have noticed, there are no Incremental or Absolute modes included in the Preparatory Commands (G codes). On a CNC turning center or Lathe, the mode is always set to Absolute and diameter, if an Incremental movement is required the letters U or W are used for X or Z respectively.

X OVERCUT

Most lathe tools have a radius on the front or cutting edge; it is referred to as Tool Nose Radius. This radius must be compensated for in the calculation of the tool path much like the tool radius offset in milling operations, this offset is known as Tool Nose Radius Compensation. We will discuss this later on in this book but for now, know that when we program a facing operation we must account for the radius in our final X position of the facing move to create a flat surface. This extra value that we program is sometimes referred to as **overcut.**



COMMONLY USED MISCELLANEOUS **M** CODES

CODE	FUNCTION
M00	The M00 code is used for a Program Stop command on the machine. It stops the spindle, turns off coolant and stops look-a-head processing. Pressing CYCLE START again will continue the program on the next block of the program.
M01	The M01 code is used for an Optional Program Stop command. Pressing the OPT STOP key on the control panel signals the machine to perform a stop command when the control reads an M01 command. It will then perform like an M00. Optional stops are useful when machining the first part to allow for inspection of the part as it is machined.
M03	Starts the spindle CLOCKWISE for most machining. Must have a spindle speed defined. The M03 is used to turn the spindle on at the beginning of program or after a tool change.
M04	Starts the spindle COUNTERCLOCKWISE. Must have a spindle speed defined.
M05	STOPS the spindle. If the coolant is on, the M05 will turn it off.
M08	Coolant ON command.
M09	Coolant OFF command.
M10	Open Chuck
M11	Close Chuck
M12	Tailstock Quill IN
M13	Tailstock Quill OUT
M17	Turret Indexing Forward
M18	Turret Indexing Reverse
M19	Spindle Orientation
M21	Tailstock Forward
M22	Tailstock Backward
M23	Thread Gradual pullout ON
M24	Thread Gradual pullout OFF
M30	Program End and Reset to the beginning of program.
M41	Low Gear selection
M42	Medium Gear selection 1
M43	Medium Gear selection 2
M44	High Gear selection

NOTE: On the Haas lathe only one "M" code can be used in each block. The "M" codes will be the last command executed in a line, regardless of where it's located in that block.

RAPID **G00** AND LINEAR **G01** INTERPOLATION

G00 RAPID TRAVERSE

This code is used for rapid motion of the tool in air to traverse from one position to another as fast as possible. This code will work for both axis motions at once.

This G00 code is modal and causes all the following blocks to be in rapid (up to 1000 in./min.) motion until another Group 01 code is specified, for example G01 linear interpolation.

Generally, rapid motions "will not" be in a straight line. All the axes specified are moved at the maximum speed and will not necessarily complete each axis move at the same time. It activates each axis drive motor independently of each other and, as a result, the axis with the shortest move will reach its destination first. So ***you need to be careful of any obstructions to avoid with this type of rapid move.***

- G00 is used when you are positioning the tool in 'fresh air'.
- Retracting from a hole you have drilled.
- Rapid traverse is not used when cutting the part.
- Used incorrectly, rapid traverse will break a tool very easily and possibly remove the part from the chuck.

G01 LINEAR INTERPOLATION

This G code provides for straight line (linear) motion with programmed feedrate for all axis motions from point to point. Motion can occur with both axes at once.

All axes specified will start at the same time and proceed to their destination and arrive simultaneously at the specified feedrate.

To program a feedrate, the F command is used. The F command is modal and may be specified in a previous block.

G01 is used for

- Drilling a hole
- Turning a diameter
- Machining a profile I.D and O.D.
- Grooving I.D and O.D.

EXAMPLE OF PROGRAM **START-UP BLOCKS**

N10 G18 G20 G40 G54 G80 G97 G99

Many programs have a G code default line or **"Safety Block"** at the beginning of the program, this is to ensure the machine control is in a safe start condition before proceeding with the program. If the previous program had failed to cancel certain function this **"Safety Block"** would help ensure that the new program starts with the appropriate settings.

For example the G codes above would perform the following at the start of the program:

G18 - ZX Plane Selection
G20 - Inch Programming
G40 - Cancel Tool Nose Compensation
G54 - Work Offset Command
G80 - Cancels Canned Cycles
G97 - Constant Non-Varying Spindle Speed
G99 - Feed Per Revolution

MACHINE DEFAULTS

When the machine tool is powered on the control automatically recognizes a series of codes. On the Haas lathe the G codes listed below are set when the lathe is powered up:

G00 Rapid Traverse

G18 XZ Circular Plane Selection

G40 Cutter Compensation Cancel

G54 Work Coordinate Zero #1

G64 Exact Stop Cancel

G80 Canned Cycle Cancel

G97 Constant Surface Speed Cancel

G99 Feed Per Revolution

EXAMPLE OF PROGRAM **START-UP BLOCKS**

<table>
<tr><td>%</td><td colspan="2">Programs must begin and end with “%”
(depending on the type of control.)</td></tr>
<tr><td>O00023</td><td colspan="2">Letter “O” and up to a five digit program number.</td></tr>
<tr><td>N10 G18 G20 G40</td><td>Nnn - Sequence Number
G18 - ZX Plane Selection
G20 - Verify Inch
G40 – Tool Nose Radius Compensation Cancel</td><td rowspan="2">Safety Startup Block</td></tr>
<tr><td>N20 G80 G97 G99</td><td>G80 - Canned Cycle Cancel
G97 - Constant Non-Varying Spindle Speed
G99 - Feed Per Revolution</td></tr>
<tr><td>N30 T0100 M41</td><td colspan="2">T0100 - Tool number #1 to be loaded into the cutting position <u>with no offset call</u>.
M41 – Select low gear if required</td></tr>
<tr><td>N40 G97 S500 M03</td><td colspan="2">G97 - constant surface speed off / revolution per minute on spindle speed is set to 500 RPM
M03 - Starts the spindle in a clockwise direction</td></tr>
<tr><td>N50 G00 G54 G41 X2.0 Z0.3 T0101 M08</td><td colspan="2">G00 – Rapid feed engagement.
G54 - Select Coordinate System #1
G41 – Tool nose radius compensation to the left of the programmed tool path.
X2.0 – Tool will rapid to a position of 1.0 units from center line of part.
Z0.3 – Tool will rapid to a position 0.3 units from finished face of part (finished face of part is usually set to Z0).
T0101 – Confirms tool #1 and assigns offset #1
M08 – Start coolant pump</td></tr>
<tr><td>N60 G96 S450</td><td colspan="2">G96 - Constant surface speed on.   Spindle will turn at S450 surface feet per minute.   Surface speed is determined by adjusting the spindle speed based on the radius of cut.
S450 - Cutting speed selection of 450 ft/min.</td></tr>
</table>

EXAMPLE OF PROGRAM **ENDING BLOCKS**

<table>
<tr><td>N200 G00 U0.05 W0.05</td><td colspan="2">G00 - Rapid Traverse
U0.05 – Rapids tool 0.05 incrementally above last X position
W0.05 – Rapids tool 0.05 incrementally away from last Z position</td></tr>
<tr><td>N210 M05</td><td colspan="2">M05 – Turn off spindle</td></tr>
<tr><td>N220 G28 U0.</td><td>G28 - Machine Zero Return
U0 - X axis in the up direction to machine zero</td><td rowspan="2">Send to machine zero X-axis first to avoid any crash.</td></tr>
<tr><td>N230 G28 W0.</td><td>G28 - Machine Zero Return
W0 - Z axis to machine zero</td></tr>
<tr><td>N240 M30</td><td colspan="2">M30 – End of Program and Reset</td></tr>
</table>

Note:
Depending on the setup of the lathe and to avoid any crashes while returning to machine zero it is usually best to move to machine zero in only one axis first. You need to be aware of where the tool is located and on its journey to machine zero, will it collide with anything?

MOVING TO MACHINE ZERO – G28 and G53

G28 - FANUC RETURN TO MACHINE ZERO THROUGH REFERENCE POINT

The G28 code is used to return to the machine zero position on all axes. If you program G28 on its own the machine will move in the X and Z axis simultaneously to machine zero.

To command only the turret to return to machine zero, and not the tailstock **if one is being used**, program in **G28 U0 W0** to send only the X and Z axes home and the tailstock will remain in place.

G00 G28 X3.00 Z2.00

The above command would position the tool from the current position to the absolute position of X3.00 and Z2.00, and then to the X and Y axis machine zero point. This is a movement through an absolute intermediate point. This can be used as a way to move to a clearance point and then return to machine zero.

Haas has a **G53** program code that works similar to G28.

Example:

G53 G00 X-2.0 Z-4.0

The machine will move negative 2.0 inches from the machine zero position in the X axis, and negative 4.0 inches from home position in the Z axis.

G53 G00 X0 Z0

This block will send the turret to machine Zero.

G40, G41, & G42 TOOL NOSE COMPENSATION

When a program is created it is done so using the insert's **command or reference** point (see figure below). **Tool Nose Compensation** is used to offset the tool by a distance that will bring the cutting edge of the insert to the proper position in relation to the specific radius of the insert being used. The radius of the tool must be input into the controller and it will calculate the proper offset known as Tool Nose Compensation.

G40 TOOL COMPENSATION CANCEL

G40 will cancel the G41 or G42 tool compensation commands that are in effect at the time.

G41 TOOL COMPENSATION LEFT (BORING)

G41 will select tool compensation to the **LEFT** of the contouring direction; generally G41 is used for boring. The tool is compensated for the radius of the tool tip. The value of the compensation (tool radius) must be entered in the controller registry during set-up.

G42 TOOL COMPENSATION RIGHT (TURNING)

G42 will select tool compensation to the **RIGHT** of the contouring direction; generally G42 is used for turning. The tool is compensated for the radius of the tool tip. The value of the compensation (tool radius) must be entered in the controller registry during set-up.