Every G Code for Your CNC [The Complete List]

December 10, 2022 by Brandon Fowler

G code, sometimes called g-code, is a simple programming language for CNC machines.

Listed below are the codes that use the G word. **These codes are preparatory commands.**

This means that they set modes or prepare the machine for various actions.

We have put together a complete list of G codes for your reference.

Not all codes are used often.

Below is a list of G codes. They have been organized into group types based on how the machine reacts to the individual code.

If you are new to G code, you should review the code groups and compare the codes within a single group.

You will quickly see they handle the same function, such as one code to turn something on and one code to turn something off.

This makes learning the codes twice as easy.

Table of Contents

CNC Movement and Travel

G00 - Rapid Travel

Moves the CNC around at the maximum possible speed. Often called rapid traverse.

Learn more about the G00 code

G01 - Linear Interpolation

Move the CNC in a straight line to a specific location. The speed of this movement is determined by the feed rate (F).

Learn more about the G01 code

G02 - Circular Interpolation, Clockwise

Move the CNC to specific location along along a specified arc or radius traveling clockwise. The speed of this movement is determined by the feed rate (F).

Learn more about the G02 code

G03 - Circular Interpolation, Counterclockwise

Move the CNC to a specific location along a specified arc or radius traveling counterclockwise. The speed of this movement is determined by the feed rate (F).

Learn more about the G03 code

G04 - Dwell

Tells the CNC not to move for a certain amount of time.

Learn more about the G04 code

Plane Selection

G17 - XY Plane Selection

The vast majority of movement in CNC programs will be relative to the XY plane. Plane selection affects how G02, G03, cutter compensation and some canned cycles are handled.

G18 - XZ Plane Selection

Plane selection affects how G02, G03, cutter compensation and some canned cycles are handled. G18 tells the machine you are machining radii around the Y axis.

G19 - YZ Plane Selection

Plane selection affects how G02, G03, cutter compensation and some canned cycles are handled. G19 tells the machine you are machining radii around the Z axis.

Unit Modes

G20 - Inch Mode

All size and location commands will be in inches until changed.

Learn more about the G20 code

G21 - Metric Mode

All size and location commands will be in millimeters until changed.

Learn more about the G21 code

Compensation Codes

G40 - Cancel Cutter Compensation

Turns off any cutter compensation modes (G41 or G42).

Learn more about the G40 code

G41- Cutter Compensation Left

Shifts the machine tool path to the left based on the radius of the cutting tool.

Learn more about the G41 code

G42 - Cutter Compensation Right

Shifts the machine tool path to the right based on the radius of the cutting tool.

Learn more about the G42 code

G43 - Tool Length Compensation

Adjusts a tool's location based on a specified tool height (H) offset.

Learn more about the G43 code

G49 - Tool Length Compensation Cancel

Turns off the tool length compensation set by a G43 or G44 command.

Learn more about the G49 code

Work Offsets

G54 - Work Offset 1

Sets a specific location in the CNC as the zero location for the program.

Learn more about the G54 code

G55 - Work Offset 2

Sets a specific location in the CNC as the zero location for the program.

Learn more about the G55 code

G56 - Work Offset 3

Sets a specific location in the CNC as the zero location for the program.

Learn more about the G56 code

G57 - Work Offset 4

Sets a specific location in the CNC as the zero location for the program.

Learn more about the G57 code

G58 - Work Offset 5

Sets a specific location in the CNC as the zero location for the program.

Learn more about the G58 code

G59 - Work Offset 6

Sets a specific location in the CNC as the zero location for the program.

Learn more about the G59 code

Canned Cycles

G73 - High-Speed Peck Drilling Canned Cycle

Peck drill a hole by repeatedly drilling, then retracting a small amount, then drilling deeper. Often used to break up chips when drilling.

Learn more about the G73 code

G74 - Left-Hand Tapping Canned Cycle

A tapping cycle for left-hand or reverse threads. The spindle must be running in reverse during this cycle. The machine will feed to the bottom of the hole and then reverse the spindle direction and back out of the hole.

Learn more about the G74 code

G76 - Fine Boring Canned Cycle

Bores a hole then stops the spindle, retracts the tool from the surface and rapid retracts the tool out of the hole.

G81 - Standard Drilling Cycle

Drills a hole then rapid retracts out of the hole.

Learn more about the G81 code

G82 - Standard Drill with Dwell

Drills a hole, dwells at the bottom of the hole for a set amount of time and then rapid retracts out of the hole.

Learn more about the G82 code

G83 - Deep Hole Peck Drilling Cycle

The machine will drill in pecks while rapid retracting out of the hole between each peck to help clear chips from the drill.

Learn more about the G83 code

G84 - Right-Hand Tapping Cycle

The standard tapping cycle with the spindle running clockwise. The CNC will feed to the bottom of the hole and then reverse the spindle direction and back out of the hole.

Learn more about the G84 code

G85 - Reaming Cycle

The machine will feed to the bottom of the hole and then rapid retract out.

Learn more about the G85 code

G86 - Boring Cycle

The machine will feed to the bottom of the hole and then rapid retract out. The tool is not pulled away from the surface which can leave a tool mark on the hole.

Learn more about the G86 code

G87 - Back Boring Cycle

Bores a hole from bottom to top. This is the reverse direction of the G86 code.

G88 - Boring Cycle with Dwell

Bores a hole and then dwells at the bottom for a set amount of time.

G89 - Back Boring Cycle with Dwell

The machine will feed to the bottom of the hole, pause for a set amount of time and then rapid retract out. The tool is not pulled away from the surface which can leave a tool mark on the hole.

Cancel Codes

G50 - Cancel Scaling

Turns off scaling mode.

G80 - Canned Cycle Cancel

Cancels any active canned cycles for drilling, boring, tapping, etc.

Learn more about the G80 code

Positioning Modes

G90 - Absolute Mode

All movement locations are relative to a fixed zero such as the work offset (G54, G55, etc.) or machine home location.

Learn more about the G90 code

G91 - Incremental Mode

All movement locations are relative to the machine's current position.

Learn more about the G91 code

Speeds and Feeds

G94 - Feed per Minute Mode

Sets the feed rate to units (inches/mm) per minute. The tool will move at a set rate.

G95 - Feed per Revolution Mode

Sets the feed rate to units (mm/inches) per revolution. The tool will move a set rate for every spindle rotation.

G96 - Constant Surface Speed

The spindle speed will vary to maintain the same surface speed of the material.

G97 - Constant Spindle Speed

The spindle will operate at a consistent RPM.

Plane Return

G98 - Return to Initial Plane

After a canned cycle the tool will retract to the start of the canned cycle.

Learn more about the G98 code

G99 - Return to Rapid Plane

After a canned cycle the tool will retract to a specified location. Can be useful for avoiding workholding fixtures.

Learn more about the G99 code

Lesser Used G Codes

G09 - Exact Stop Check (Non-modal)

Causes the machine to come to a stop after each command. Useful for making sharp corners. It causes the machine to stop after executing the current line.

G10 - Programmed Offset Input

Adjusts offset values. Can change work offset, tool length offset and/or cutter compensation offsets.

G22 - Stored Stroke Limit

Turns on a set safety zone that will cause the machine to alarm out if it enters.

G23 - Stored Stroke Limit Cancel

Turns off the stored safety zone.

G27 - Zero Return Check

Rarely used. This command checks to see that the machine has moved to the zero return position.

G28 - Zero Return

The machine will move to a specified location and then to the zero return (home) position.

Learn more about the G28 code

G29 - Return From Reference Position

The machine will move to the position referenced in the last G28 command. It will then move to the location referenced with the G29 command.

G30 - Second Position Zero Return

Similar to G28 the machine will move to a specified location and then will move to a secondary reference point in the machine such as a tool changer location.

G31 - Skip Function

Often used with probing, it is used to move the probe until it hits an object.

G44 - Negative Tool Length Compensation

Rarely used. Adjusts a tool's location based on a specified height (H) offset in the opposite of the normal direction.

G45 - Single Offset Increase

Rarely used.

G46 - Single Offset Decrease

Rarely used.

G47 - Double Offset Increase

Rarely used.

G48 - Double Offset Decrease

Rarely used.

G51 - Scaling

Scales the machined part size by a scaling factor.

Scaling factors >1 make it bigger. Scaling factors <1 make it smaller.

G52 - Temporarily Shift Program Zero

Shifts the program zero location based on a location specified with the command.

G53 - Return to Machine Zero Position

Rapid moves the CNC to the machine reference position.

G60 - Single Direction Move

Forces the machine to approach each XY location from the same direction. Helps eliminate location errors caused by machine backlash.

G61 - Exact Stop Check (Modal)

The machine will come to a stop after each line of code.

G64 - Normal Cutting Mode

Cancels both G60 and G61 modes.

G65 - Custom Macro Call

A macro is similar to a subprogram but it allows you to use variables in the program.

G66 - Custom Macro Modal Call

Makes the machine call a macro, similar to G65, with every new location given until turned off.

G67 - Cancel Custom Macro Modal Call

Cancels any active custom macro modal calls.

G68 - Coordinate Rotation Mode

Rotates the machine coordinates at a given angle around a specified location.

G69 - Cancel Coordinate Rotation Mode

Turns off coordinate rotation mode.

G92 - Program a Work Offset

Sets a work offset based on a specific location in the machine.

Frequently asked questions

How many G codes are there?

We listed 70 G codes in our list alone but in reality there are many more.

Some will have different meanings on lathes vs mills. Also, different CNC controller manufacturers will include their own special codes. For example, some will have codes such as G103 or they will switch the meaning of a code.

You should always know how your individual machine will react to a specific code.

What other types of codes are used in CNC programming?

There are numerous codes other than G codes used in CNC programming.

There are codes for locations, speeds, feeds and machine operations among others. For the most common codes, review our posts on CNC codes and M codes.

Who needs to know G code?

Anyone who works with a CNC machine would benefit from understanding G code.

While it may not be a requirement for their job depending on where they work, these positions would benefit from understanding G codes:

- CNC operator
- CNC setup
- CNC programming
- CNC machinist

Is learning G code hard?

Learning G code can be a daunting task when there are so many codes to understand.

While it is true that there are a lot of codes that can be used, you should know that most shops will only use a small selection of codes which cuts down the number you need to understand before you are up to speed.

Don't be overwhelmed. Figure out which codes are the most commonly used ones and start your learning there.

CNC Training
Geometric Dimensioning and Tolerancing Font – GD&T Keyboard Shortcuts
Best Engraving Pens for Every Budget and Material

Leave a Comment

Name *

Email *

Website

Save my name, email, and website in this browser for the next time I comment.

Post Comment

PH +1 603 287 8259 | 21 DURHAM AVE CLAREMONT, NH 03743

© 2023 MACHINIST GUIDES