G-code Explained | List of Most Important G-code Commands

by Maciej | Updated on 1 Jun 2022

If you run a 3-axis machine with CNC machine or 3D printer, then G-code is the language that tells your machine what to do. In this tutorial, we'll show the basics of the G-code language and list the most important G-code commands. We'll also show a few G-code examples that they work.

My name is Maciej, I write tutorials and I'm a hobbyist maker. I'm learning electronics and programming, and I like to share what I learn with others.

Follow me on:

Subscribe to my newsletter:

Subscribe to RSS feed

Table of Contents:

1. What is G-code?
2. How to read G-code Commands?
3. The most important Common G-code Commands:
   - G00 - Rapid Positioning
   - G01 - Linear Interpolation
   - G02 - Circular Interpolation Clockwise
   - G03 - Circular Interpolation Counterclockwise
   - G20/ G21 - Units Selection
   - G53/ G54/ G55 - G-code Plane Selection
   - G90/ G91 - Positioning G-code commands
   - More Commands and Rules
4. Simple G-code Program Example
5. Conclusion

Subscribe to our newsletter

Stay up to date with our latest projects, tutorials, and exclusive content!

Enter your email address

Subscribe

Join 18,000+ Subscribers

Latest Posts

Popular Posts

What is G-code?

G-code (or geometric language 1) is a computer numerical control language, in computer science, is the "numerical control" (NC) programming language. It is used for computer-aided manufacturing (CAM) and is the language in which computer-aided machines (lathes, mills, 3D printers, etc.) are controlled. It is a flexible and extensible language, in which the machine tool and its path are defined.

In case of a machine tool such as a lathe or mill, the cutting tool is driven by these instructions, which are typically created automatically by software called a CAM (Computer Aided Manufacturing) package.

Similarly, in case of additive manufacturing or 3D printers, the G-code commands the printer what to do. Most slicer software has an option that allows you to generate G-code.

How to read G-code Commands?

In the past years a lot of code was written for machine tools, so the G-code is well established. It is also a very old language.

```
%
O0001 (PROGRAM NAME - EXAMPLE)
(DATE=DD-MM-YY - 01-01-00 TIME=HH:MM - 12:00)
N100 G00 G54 G17 G40 G49 G80 G90
(T1 | 6 FLUTE CARBIDE END MILL)
N102 G43 H1 Z1.
N104 X-0.5 Y-0.5 S3500 M03
N106 G01 Z-0.125 F20. M08
N108 X0.0 Y-0.5
N110 X0.0 Y0.0
N112 X-0.5 Y0.0
N114 X-0.5 Y-0.5
N116 G00 Z1.
N118 M09
N120 G91 G28 Z0. M05
N122 G28 X0. Y0.
N124 G90 G54 G17 G40 G49 G80 G90
N126 M30
%
```

A typical G-code program looks like this:

```
%
O0001 (PROGRAM NAME - EXAMPLE)
(DATE=DD-MM-YY - 01-01-00 TIME=HH:MM - 12:00)
N100 G00 G54 G17 G40 G49 G80 G90
(T1 | 6 FLUTE CARBIDE END MILL)
N102 G43 H1 Z1.
N104 X-0.5 Y-0.5 S3500 M03
N106 G01 Z-0.125 F20. M08
N108 X0.0 Y-0.5
N110 X0.0 Y0.0
N112 X-0.5 Y0.0
N114 X-0.5 Y-0.5
N116 G00 Z1.
N118 M09
N120 G91 G28 Z0. M05
N122 G28 X0. Y0.
N124 G90 G54 G17 G40 G49 G80 G90
N126 M30
%
```

This is the following meaning:

- % - Start of file
- Every line **N** is code command and ends with the **EOB** (end-of-block) character (**LF** in ASCII, or **CR+LF**)
- **N** is the line number (optional) in the range from **N1** to **N 99999**, (some machines accept more than 99999 lines, but it is not standard)
- **O** is the program number (in the example **O1**)
- **G** is the command (in the example **G00**, **G54**, **G17**, **G40**, **G49**, **G80**, **G90**)
- **X**, **Y**, **Z** are the coordinates (in the example **X-0.5**, **Y-0.5**, **Z1.**)
- **S** is the spindle speed (in the example **S3500**)
- **F** is the feed rate (in the example **F20.**)
- **M** is the miscellaneous function (in the example **M03**, **M08**, **M09**, **M05**, **M30**)

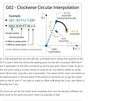
In step **N**, the **G54** (**G17**, **G40**, **G49**, **G80**, **G90**) command sets the machine to a known state. The **G43** command activates the tool length offset, and the **H1** specifies the offset number. The **M03** command turns on the spindle, and the **M08** activates the coolant. The **G00** command moves the tool to the specified coordinates, and the **G01** command performs a linear interpolation with a specified feed rate. The **M09** command turns off the coolant, and the **G91 G28 Z0. M05** command moves the tool to the Z-axis home position and stops the spindle. The **G28 X0. Y0.** command moves the tool to the X-Y home position, and the **G90 G54 G17 G40 G49 G80 G90** command resets the machine to its default state. The **M30** command ends the program and resets the machine to its initial state.

The most Important Common G-code Commands:

Any code an slicer/cnc tool uses to make 3D or 2D cut or move can be written using a few G-code commands. Most of the time, you don't need to know all of the G-codes, only through some basic commands, since most of the codes are set up by the CAM software. Even if you want to write G-code manually, it is easy. Here is a few G-codes and what they do:

G00 - Rapid Positioning

The **G00** command moves the machine to a specified location. It is used to move the tool quickly from one location to another, without cutting any material. The **G00** command takes the X, Y, and Z coordinates as parameters, and the machine will move to the specified coordinates at the maximum speed. The **G00** command is useful for quickly moving the tool from one location to another, and it is often used in conjunction with other G-codes, such as the **G01** command, to perform more complex operations.



The G00 is mostly used for positioning, and it is not suitable for cutting material, because the machine is programmed to move in a straight line, regardless of any obstacles or boundaries. The G00 command is a powerful tool for quickly and efficiently moving the tool from one location to another, but it should be used with caution and in conjunction with other G-codes to ensure that the machine does not damage the workpiece or the tool.

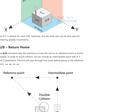
G01 – Linear Interpolation

The **G01** command performs a linear interpolation, which means it is used to move the tool in a straight line from one location to another at a specified feed rate. The G01 command takes the X, Y, and Z coordinates as parameters, and the machine will move the tool to the specified coordinates at the specified feed rate. The machine controller calculates intermediate coordinates between the current position and the specified coordinates, and it adjusts the speed of the tool accordingly to maintain the specified feed rate. The G01 command is useful for performing cuts and other operations that require a specific feed rate.

Unlike the G00 command which is used for positioning the tool, the G01 command is used for cutting material. The G01 command can be used to create a variety of shapes and patterns by moving the tool in a straight line, and it can also be used to create angled cuts and other more complex operations.

G02 – Circular Interpolation Clockwise

The **G02** command tells the machine to move clockwise in a circular pattern. It takes the X, Y, and Z coordinates as parameters, and the machine will move the tool to the specified coordinates in a clockwise direction. The G02 command also takes the I and J parameters, which specify the center of the circular pattern. The I and J parameters are relative to the current position of the tool. The tool will move in a circle with the center at the specified I and J coordinates.



The G02 command can be used to create a variety of circular patterns, such as circles, arcs, and spirals. The G02 command is useful for performing cuts and other operations that require a circular motion, such as milling a circular pocket or cutting a circular groove in a workpiece.

G03 – Circular Interpolation Counterclockwise

The **G03** command tells the machine to move counterclockwise in a circular pattern. It takes the X, Y, and Z coordinates as parameters, and the machine will move the tool to the specified coordinates in a counterclockwise direction. The G03 command also takes the I and J parameters, which specify the center of the circular pattern. The I and J parameters are relative to the current position of the tool. The tool will move in a circle with the center at the specified I and J coordinates.

The G03 command can be used to create a variety of circular patterns, such as circles, arcs, and spirals. The G03 command is useful for performing cuts and other operations that require a circular motion, such as milling a circular pocket or cutting a circular groove in a workpiece.

G20/ G21 – Units Selection

The G20 and G21 commands allow you to set units, either inches or millimeters.

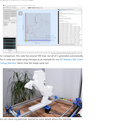
- G20 – inches
- G21 – millimeters

The default mode depends on the control, but at the beginning of the program, it has good sense to set the machine to the unit system, the software and/or the previous program used.

G53/ G54/ G55 – G-code Plane Selection

Which G-code commands are called the most depends on the machine.

- G53 – off-plane
- G54 – XY plane
- G55 – YZ plane



The G53, G54 and G55 commands set the plane that the machine will use for cutting and moving.

G20 – Return Home

The **G28** command tells the machine to return home (or to reference point or to home position) in order to avoid collision, as can include an intermediate point and X Y Z coordinates. The machine will move to the specified coordinates.

The G28 command can be left blank and the machine will go to home.

G90/ G91 – Positioning G-code commands

While the G00 and G01 commands can set the machine how to move from point to point, the G90 and G91 commands set how the machine will move.

In absolute mode (programming G90) the machine will move from point to point, as it is, for the command (or, in case of G01 the feed rate to move from point to point). In relative mode (programming G91), the programming of the next command or the next point, is the distance to move from the current position. For example, if the current position is at X=10, the next command is set to move to X=20, the machine will move to point X=30. The command can be a G00 rapid move or a G01 linear move.

Absolute Mode and Relative Mode

More Commands and Rules

Most of the G-code commands are short and cryptic, but the most common ones have more descriptive names. Here are some examples (most compatible with Mach3 and other popular CNC controllers):

In addition to the G-codes, there are the M-code commands which are used when processing a specific Program or a job program. Here are the common M-code commands:

- M00 - Program stop
- M01 - Stop if enabled
- M03 - SPINDLE ON - clockwise
- M04 - SPINDLE ON - counterclockwise
- M05 - Spindle stop
- M06 - Tool change
- M08 - Flood coolant ON
- M09 - Flood coolant OFF
- M30 - End of program

In case of a G-program:

- F - Feed rate
- S - Spindle speed setting
- T - Tool number selection
- X - X-axis position
- Y - Y-axis position, Y-axis offset, Y-axis rotation
- Z - Z-axis position

Some of these commands want appropriate parameters. For example, when turning on the spindle, you need to set the speed. The parameter is a number following the letter. For example, you can turn on the spindle at speed of 1000 RPM:

We can also make their every code **modal** which means they remain in effect until they are cancelled or replaced by another command. For example, if we set the feed rate using a command G1 F 10, then the machine will keep it at 10 for all the following G1 commands until we set a different feed rate in a command, or cancel it, for example, with the command G0 F0.

```
G1 F10
G1 X10 Y10
G1 X20 Y20
```

Is this code good for the high-end operations? We don't want to restate it every time, it is easier to keep it in the memory.

In order to make the code more clear, we have the **G90** in front of the commands. The G-code is modal, so everything that is set in a program, from now on, will be in the absolute mode.

Simple G-code Program Example

Nevertheless, with modern CAM, the code can be very complicated. Here is a very simple G-code. Here is an example:

```
%
O0001 (PROGRAM NAME - EXAMPLE)
(DATE=DD-MM-YY - 01-01-00 TIME=HH:MM - 12:00)
N100 G00 G54 G17 G40 G49 G80 G90
(T1 | 6 FLUTE CARBIDE END MILL)
N102 G43 H1 Z1.
N104 X-0.5 Y-0.5 S3500 M03
N106 G01 Z-0.125 F20. M08
N108 X0.0 Y-0.5
N110 X0.0 Y0.0
N112 X-0.5 Y0.0
N114 X-0.5 Y-0.5
N116 G00 Z1.
N118 M09
N120 G91 G28 Z0. M05
N122 G28 X0. Y0.
N124 G90 G54 G17 G40 G49 G80 G90
N126 M30
%
```

Description of the G-code program:

- Code explanation: The sequence %O... is always present at the beginning and at the end of the program.
- Line numbers: Line numbering is optional, but it is recommended to use. It starts, typically, at the beginning of the program, with the sequence number N100.
- Program beginning: G90 G54 G17 G40 G49 G80 G90
- Tool change: Tool change is performed using the command G43 H1 Z1.
- Coolant: Coolant is activated by M08.
- Spindle: Spindle is activated by M03 S3500.
- Linear cutting: A linear feed G01 Z-0.125 F20. is used.
- Tool path: The tool path is defined by X0.0 Y-0.5 X0.0 Y0.0 X-0.5 Y0.0 X-0.5 Y-0.5.
- Tool return: The tool returns to the initial position Z1.
- Coolant off: Coolant is deactivated by M09.
- Spindle stop: Spindle is stopped by M05.
- Return home: The machine returns to the home position X0. Y0.
- Program end: The program ends with M30.

Here's how the code looks exactly in the cam to cut CNC machine through the workpiece (using Mach4 software):

Using Mach4 to run the G-code example and cut through the workpiece

The G-code is very simple, it only cuts a straight line. It is not suitable for any real-world cutting. But it gives you a good idea of how the G-code looks like.

The finished workpiece. Made using a CNC router and a 6 mm end mill

Conclusion

We can conclude that G-code is a flexible and powerful language that is used to control computer-aided machines such as lathes and mills and 3D printers. It is a language that is used to define the path of the machine tool and the cutting tool, and it is typically created automatically by software called a CAM package. The G-code language is a powerful tool for automating the manufacturing process and for creating complex parts with high precision.

If you found this article useful, share it with CNC and 3D printing enthusiasts. Also feel free to ask any question in the comments section below.

Tutorials

Latest Posts: Sequencer, CNC DIY Sequencer | Arduino Nano 33 Power Supply & Battery Charger | Arduino Nano 33 Power Supply

Arduino Nano 33 Power Supply & Battery Charger

6 thoughts on "G-code Explained | List of Most Important G-code Commands"

Corey Ellis
June 2, 2022 at 1:01 pm

Thank you sir!!! Exactly, epitomized in simple terms so even I can understand!

Reply

Audio
June 2, 2022 at 1:03 pm

I found this article and the comments very useful for I am learning this all on my own. Thank you!

Reply

Nicolas
June 2, 2022 at 1:03 pm

Excellent article. Coming through in a detail and little likely way a couple of things I didn't know about. Thanks for the information. Cut G00 to G03 - Speed - Speed - Speed