What is G-Code? What is M-Code? What Do They Mean in CNC Machining?

07/13/2021 | DAN KARBON AND TOM HERR

As its name clearly states, CNC (computer numerical control) is automated control of machining tools using a computer. Yet, that doesn't just happen; it requires the right **coded programmed** instruction.

So, what is G-code? What is M-code? And how do **G-code** and **M-code** programming actually work? This article explains each one, but first let's touch on what programming actually is.

A **program** is a sequence of codes and data that tells the CNC machine what to do. The programmed codes, along with the right tooling in a CNC machine center, allow for **correct** and **repeatable** part manufacturing.

If you're interested in the programming process, [read this article](#).

As a CNC machine runs, the code is displayed on the operator's monitor and is seen scrolling along, being read and acted upon by the machine.

What is G-Code — The Master of Action

Simply put, G-code is a software programming language used to **control a CNC machine**. And, even with complex CNC machining, G-code is written in a straightforward, **logical** way.

A "**G**" is followed by a number, which is a command to change geometry. For instance, "G00" is a command for rapid movement. It moves the tool to part geometry at the maximum rate and is generally used to move the tool and part from near to far or vice versa.

Here are some examples of G-code:

- G00: Rapid move to specific coordinate position

- G01: Linear feed move

- G02/G03: Clockwise/counterclockwise feed move

These codes, such as G00, are followed by a **geometric location**. The geometry is conveyed with coordinates on axes, so an "**X**" coordinate and a "**Y**" coordinate on a plane.

Let's break down "G00G58X-120.Y-5." You know the "G00" means a rapid move. "G58" specifies **work coordinates**, changing geometry relative to work offsets, like part origin. "X-120." means to go to "-120mm" on the X axis, and "Y-5." means to go to "-5mm" on the Y axis.

G-code can repeat an action indefinitely until it is stopped. For example, G01 followed by a sequence of X, Y, or Z positions, means all linear feed moves occur at G01. A typical machining example is **face milling**, where the sequence of positions is used to machine the part's face contour.

All of these coordinates in G-code allows the CNC machine to **consistently** perform and also be flexible enough to produce **different** parts.
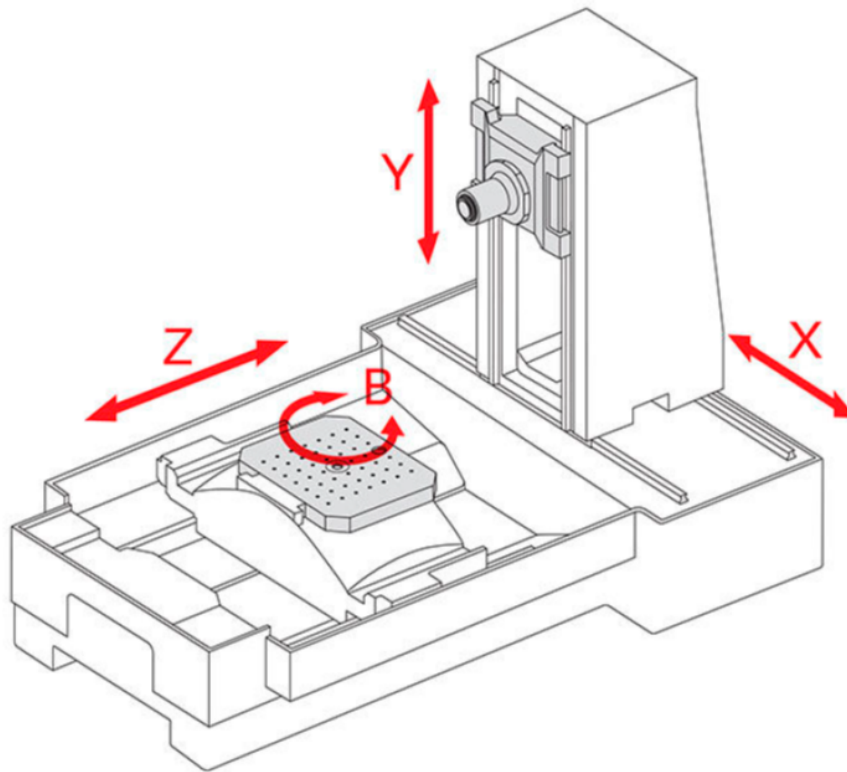
The X and Y axes position the spindle precisely to perform a cut or other function.

What is M-Code — The Organizer of Functions

M-code controls **miscellaneous functions**, and typically these can be thought of as non-geometry machine functions. Examples include spindle rotation start and stop, coolant on and off, pallet change, etc. Some of these functions may vary by the **specific machine**.

Here are some examples of M-code:

- M00: Stop whatever function the program is currently doing

- M03: Rotate spindle clockwise

- M08/M09: Flood coolant on/off

Similar to G-code, M-code consists of an "**M**" and a number. Unlike G-code, however, M-code can be more specific for different machines and is more **customizable**. Both types are needed to have a CNC machine perform its functions.

Like any language, CNC has many other aspects to it. This chart is just a sampling of all the **program lines** that could be included in the code.

Program Line	Explanation
N102M6T2	Block number 102, Tool change to tool 2
(3" FACEMILL)	Comment - not read by machine, but tells operator engineer tool 2 is a 3 inch facemill
M82	Pallet A Check
IF[#10002+#11002LT150.]GOTO2001	Tool height min check
IF[#10002+#11002GT154.]GOTO2002	Tool height max check
G00G90B267.	Rapid, absolute positioning, Pallet rotates to B 267 (degrees)
M08	Coolant on
(PART B)	Comment

Some examples of code programming lines and what they mean. Anything in parentheses is not read by the machine. These are comments for the programmer and operator.

Other Commands. Other Machines.

In addition to G-codes, M-codes, and the X/Y/Z axes positioning, other commands are used in CNC machining. An M03 spindle rotation start command needs an "S" command, which controls **spindle speed** in rpm.

Here's how that gets added to our current code sample: G00G58X-120.Y-5.M03S3820.

The "S3820" indicates a spindle speed of **3820 rpm**. So far, the program has set the geometry and the spindle speed.

Other possible functions:

- F: Feed rate

- T: Tool number

- H: Tool length offset call up

Many other alphanumeric designations can be used, depending on the CNC machine's capabilities.

Keep in mind, the specific G-code and M-code examples we've explored are based on **horizontal** and **vertical machining center** programs. CNC lathes use G-code and M-code too, but you can expect some different commands and sequences for turning.

The Power of Software

While machines are frequently programmed directly with G-code, a CNC programmer often uses a high-level **computer-aided manufacturing** (CAM) programming tool.

Mastercam is a code-producing software that helps programmers create the right code for the project. Standard in the CNC industry, Mastercam takes input from the **programmer** (everything it needs to do), and it produces the right G-code and M-code programming.

Here's a simplified example of how Mastercam works. The programmer imports a machine **model** and the machining **fixture**. Then, the programmer selects the **tools** and the tooling paths of where the spindle will go, and the code is produced by the software. The programmer works with the CAD models, but refers to the **print** for the exact specifications and tolerances.

As powerful as Mastercam is, it could require **two weeks** of programming if the part is complex (large size and many tools needed), yet it could only take **two hours** for a simple part. Obviously, a 3-tool job is simple compared to a 100-tool job.
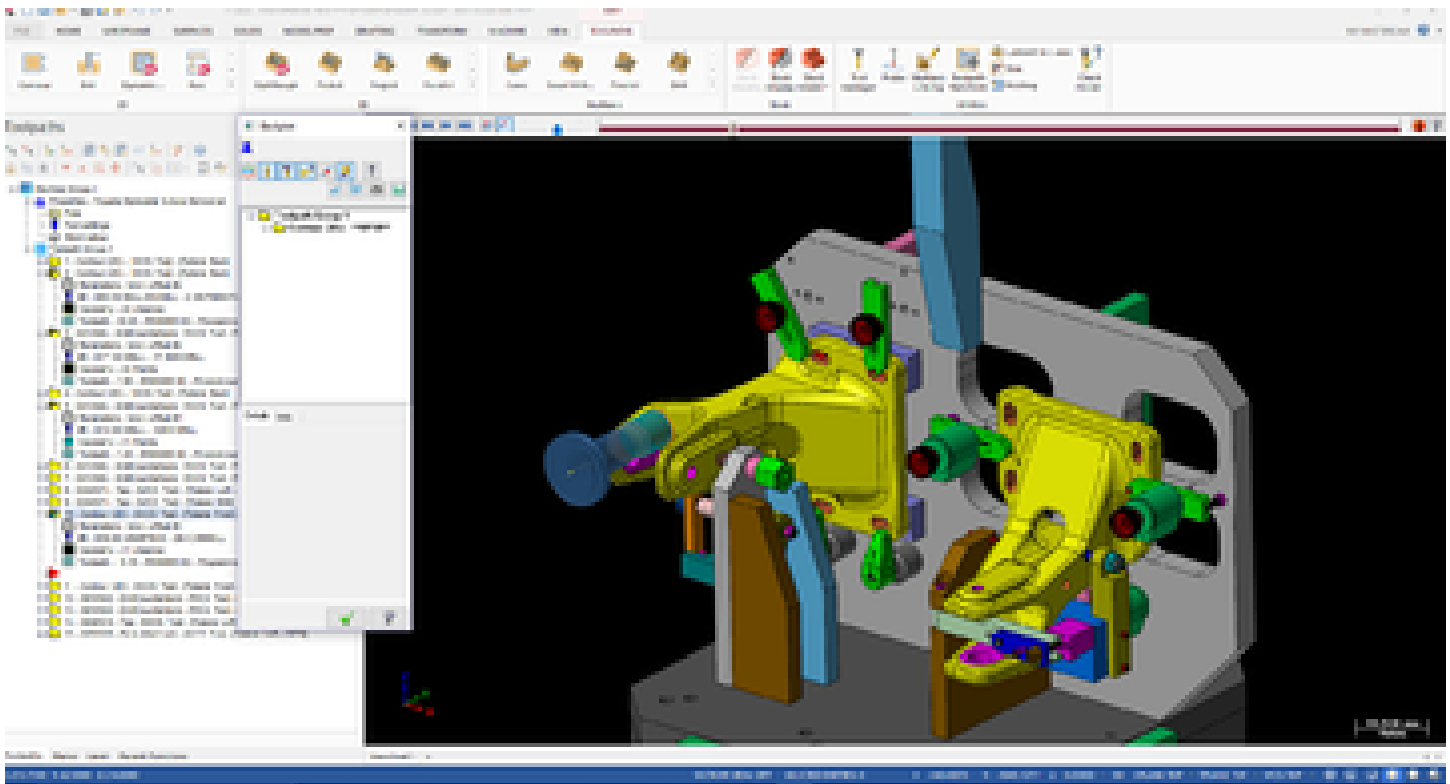
Mastercam software shows a visual of the entire project; at left are folders containing the code needed to perform various functions.

Who Needs to Know G-Code and M-Code?

Depending on your **job responsibilities**, maybe reading this article is all you need to know about CNC programming using G-code and M-code. Or, you may need to **dig deeper** to properly execute your daily duties.

Within a CNC machine shop, knowing something about code is helpful for many positions. Shops with a good internal **training program** will explain the basics of coding to new employees during the orientation process. Others may include it as part of on-the-floor training for operators.

Depending on the part and process, the CNC machine center and program run **repeatedly** with limited interaction. The operator loads, unloads, inspects, deburrs parts, and packages the part.

In a perfect world, once the correct CNC code is programmed, it's done, and **no edits** are needed on the floor. Many projects, however, require slight adjustments, like adjusting