


[Guides](#)

G-Code Cheat Sheet of Common Commands

Share: [in](#) [f](#) [t](#) [e](#)

G-code is the language used to direct [CNC machines](#). Combined with M code and a number of other letter-designated “words,” [turning](#) and [machining centers](#), as well as other CNC equipment, can be used to manufacture parts with little operator intervention. The majority of these codes are self-explanatory but some require some amplification to understand their behavior. This article will sift through some of these common G and M codes.

G-code was originally developed to control X-Y plotters and was, for all intents and purposes, a two-axis program. Despite its limitations, it is still used today for many CNC machines even as these machines have grown increasingly more sophisticated. Modern [CAM programs](#) can generate G-code behind the scenes; however, knowledge of the basics of G-code can be helpful in understanding the way in which these machines operate. G-code may still be entered manually as well through an operator interface.

The standard for G-code is EIA-274-D. Manufacturers will generally adhere to this standard but variations have developed over the years from manufacturer to manufacturer. Always consult the maker’s manual for a particular machine to ensure the correct application of the code.

More information on G and M codes may be found in other related articles in this series listed at the end of this article.

Route commands

G00 means rapid move and is used to bring cutting tools to a start position in the fastest time possible. The move may or may not be interpolated, depending on the machine. The format is G00XxYyZz.

-->Interested in [CNC Machining Services](#)? Try out Xometry’s [Instant Quoting Engine](#)SM to make your project come to life. Free shipping in the USA.

G01 means linear cutting and the tool moves to the specified coordinates at the set speed along an interpolated path. That is, for a given X, Y, and Z coordinate, the machine will move all axes simultaneously. The format is G01XxYyZz.

G02 and G03 produce clockwise and counterclockwise arcs, again interpolated. In addition to the end points of the arc, an arc center is specified. The format is G02XxYyIij where I and J are the coordinates of the center point in the X-Y plane. Moves in the other planes are similarly constructed.

Canned cycles

Canned, or fixed, programs for drilling distinguish between straight in and out, those that dwell at the hole bottom, and those that peck the hole with a series of intermediate retractions (for chip breaking). G81 is the code for the standard cycle, G82 pauses at the bottom of the hole, and G83 pecks through the hole cycle. A typical format is G83XxYyZtZfZs, where t represents the total Z travel, f, the first peck increment, and s, the subsequent peck increments. Multiple holes are drilled using a single command and separate lines for each coordinate pair, followed by G80 which cancels the drilling cycle. Retraction from the hole is automatically carried out at a rapid speed. Similar canned cycles handle boring, tapping, etc.

Routines that produce several identically tapped holes can be programmed through cut-and-paste to carry identical coordinates to chamfering, drilling, and tapping operations with only minor modifications necessary in the code.

A number of similar cycles are available as non-modal, one-shot routines that do not require an end command. The majority of G-code commands are modal – they remain in effect until another command overrides the current one. Modal commands are grouped so that more than one may appear on any line, but not from the same group. For example, the units setting commands G20 and G21 would not be used on the same line.

Absolute/Incremental coordinates

Absolute motion is the typical default setting. Moves are made relative to the machine origin or to another origin assigned by the G92 command. G91 changes the motion to incremental coordinates so that moves are made from the current position with no regard to machine origin. G90 returns the program to absolute motion.

Tool compensation

Cutter radius offsets G41 and G42 account for the geometry of the tool and permit the programming of part dimensions, for which the program determines the path the tool takes based on the tool dimensions. It allows for tools of different radii to be used by the same program. Likewise, CNC [lathes](#) make use of tool-nose radius compensation. This recognizes that turning tools only come to points in theory, but in reality, have finite dimensions at the interface between the cutting tool and the workpiece.

Setup commands

Feed rate is set with the address Fn, where n represents the feed rate/minute or feed rate/rev. Likewise, address S pertains to spindle speed. T calls a particular tool after a tool change command.

Although machine manufacturers vary their assignments of M codes, some are fairly standardized, including:

- M00 and M01 for program stop and optional program stop
- M02 End of program
- M03, M04, and M05 for spindle control
- M06 Tool change
- M07 and M08 for coolant on/off
- M30 End of the program/return to start
- M41 and M42 for spindle gear range

Other typical M codes launch and end subprograms, usually M98 for starting a subroutine and M99 for returning to the main program. Subprograms are designated with the letter P and an L specifies the number of times the subprogram should run.

Macro variables are called up with an X followed by #n, where n represents the address of the stored variable, such as #123. Macros can be used to change dimensions on part families of identical shapes but with smaller or larger sizes.

Miscellany

Individual line numbers may be called out with an N although numbering lines is not necessary. The recommended practice in numbering program lines is to increment each line by five to allow the insertion of new lines into a program without losing the sequence. Comments may be added to any line with opening and closing parentheses.

Summary

This article presented a few tips for understanding how G-code is used in CNC programs and the format for the common types of commands. For more information on related products, consult our [other guides](#) or visit the [Thomas Supplier Discovery Platform](#) to locate potential sources of supply or view details on specific products.

Other CNC Programming Articles

- [An Introduction to G-Code and CNC Programming](#)
- [List of Common G-Code Commands and What they Mean](#)
- [An Introduction to M-Code and CNC Programming](#)
- [Examples of G-Code for Common CNC Functions](#)
- [Subprograms, Macros and Parametric Programming for CNC Machining](#)

Share

Related Resources

[Daily Bite](#)
[Pentagon Spends Millions on Language for Military Drones](#)

[Daily Bite](#)
[Why Do We Think Garlic Keeps Vampires Away?](#)

[Daily Bite](#)
[Engineers, Contractors Build Sculptures Out of Canned Food](#)

[Daily Bite](#)
[Scientists Discover the Heaviest Animal Ever](#)

[Daily Bite](#)
[Fruit Flies Produce Promising Antibiotic](#)

Browse by Category

[Adhesives & Sealants](#)

[Automation & Electronics](#)

[Calculators](#)

[Chemicals](#)

[Custom Manufacturing & Fabricating](#)

[+ View More](#)

Find Suppliers, Insights, Tools and More...

Become part of North America’s largest and most active network of B2B buyers and industrial/commercial suppliers.



Select From Over 500,000
Industrial Suppliers



Receive Daily
Industry Updates



Search Over
6 Million Products



Download 2D & 3D
CAD Models

Start Sourcing Suppliers

Claim Your Company Profile →

[Find Suppliers](#)

[Supplier Discovery](#)

[Product Catalogs](#)

[CAD](#)

[Diversity](#)

[Instant Quotes](#)

[Buyer & Engineer Reviews](#)

[Industry Insights](#)

[Topics](#)

[Thomas Index \(TMX\)](#)

[Daily Bite](#)

[Guides](#)

[White Papers](#)

[Certification Glossary](#)

[Subscribe](#)

[Industry Update Podcast](#)

[For Business](#)

[Advertise](#)

[Content & Data Services](#)

[Marketing Services](#)

[Thomasnet Reviews](#)

[Claim Your Company Profile](#)

[Thomas WebTrax](#)

[Events & Webinars](#)

[Site Maps](#)

[Categories](#)

[Featured Companies](#)

[Featured Categories](#)

[Featured Products](#)

[Featured Catalogs](#)

[About](#)

[Thomas Brand Center](#)

[Careers](#)

[Press Room](#)

[Sign Up](#)

[Sign In](#)

[Contact](#)

[Help Center](#)

