

CNC CODE

In order to tell a CNC machine what to do you need to input a program or a code into it.

An universal standard for CNC machines is M-code & G-code

M- CODE

M codes relate to other functions such as coolant, speed, or tool changes.

M code is used for controlling machine part/accessories/function control. i.e. coolant on/off, spindle clockwise/anticlockwise, tool change, etc.

In computer programming, machine code is any low-level programming language, consisting of machine language instructions, which are used to control a computer's central processing unit (CPU).

Each instruction causes the CPU to perform a very specific task, such as a load, a store, a jump, or an arithmetic logic unit (ALU) operation on one or more units of data in the CPU's registers or memory.

Machine code is a strictly numerical language which is designed to run as fast as possible, and may be considered as the lowest-level representation of a compiled or assembled computer program or as a primitive and hardware-dependent programming language.

Basic M-codes:

M00 - Program stop; Mill and Lathe

M01 - Optional program stop; Lathe and Mill

M02 - Program end; Lathe and Mill

M03 - Spindle on clockwise; Lathe and Mill

M04 - Spindle on counter clockwise; Lathe and Mill

M05 - Spindle off; Lathe and Mill

M06 - Tool change; Mill

M08 - Coolant on; Lathe and Mill

M09 - Coolant off; Lathe and Mill

M Codes

- M00 Program stop
- M01 Optional program stop
- M02 Program end
- M03 Spindle on clockwise
- M04 Spindle on counterclockwise
- M05 Spindle stop
- M06 Tool change
- M08 Coolant on
- M09 Coolant off
- M10 Clamps on
- M11 Clamps off
- M30 Program stop, reset to start

G- CODE

G codes relate to the motion or positioning of the tool relative to the workpiece.

G code is used for geometry control of profile while machining. i.e., straight line, arc, for cutting and non-cutting moves of tool.

G-code is just a programming language where you command the machine where to move, how to move and what additional things to do.

G-code (also RS-274) is the most widely used computer numerical control (CNC) programming language. It is used mainly in computer-aided manufacturing to control automated machine tools, and has many variants.

G-code instructions are provided to a machine controller (industrial computer) that tells the motors where to move, how fast to move, and what path to follow.

The two most common situations are that, within a machine tool such as a lathe or mill, a cutting tool is moved according to these instructions through a toolpath cutting away material to leave only the finished workpiece and/or an unfinished workpiece is precisely positioned in any of up to nine axes around the three dimensions relative to a toolpath and, either or both can move relative to each other.

G CODE	FUNCTION
G00	RAPID POSITIONING
G01	LINEAR INTERPOLATION
G02	CIRCULAR INTERPOLATION CW
G03	CIRCULAR INTERPOLATION CCW
G04	DWELL
G20	INPUT IN INCH
G21	INPUT IN MM
G28	RETURN TO REFERENCE POINT
G40	CUTTER COMPENSATION CANCEL
G41	CUTTER COMPENSATION LEFT
G42	CUTTER COMPENSATION RIGHT
G43	TOOL LENGTH COMPENSATION (+)
G44	TOOL LENGTH COMPENSATION (-)
G49	TOOL LENGTH COMPENSATION CANCEL
G73	PECK DRILLING CYCLE
G74	COUNTER TAPPING CYCLE
G76	FINE BORING CYCLE
G80	CANNED CYCLE CANCEL
G81	DRILLING CYCLE, SPOT BORING
G82	DRILLING CYCLE, COUNTER BORING
G83	PECK DRILLING CYCLE
G84	TAPPING CYCLE
G85	BORING CYCLE
G90	ABSOLUTE COMMAND
G91	INCREMENTAL COMMAND
G92	PROGRAMMING OF ABSOLUTE ZERO POINT
G94	FEED PER MINUTE
G95	FEED PER REVOLUTION