

CNC programming

Part machining on a machine tool requires a plan of action. A sequence of motions and actions have to be carried out. This might be performed by human effort or by automation. When a computer program, on being executed, successfully guides a machine tool through the sequence of motions and actions necessary for machining a part, it is an example of computer numerical control programming.

This computer program need not necessarily have to be in any sophisticated language like C or Java. More often than not, it is in the primitive G & M code programming language which has been used right from the advent of CNC machines.

Here, the author does not wish to categorize Computer control, Computer Numerical Control and Numerical Control as separate entities in machine tool control. They are essentially the same, employing numerical data to achieve control.

G & M Code programming language is very simple to understand and make use of. It is a coded language to intimate to the machine the type of motion to execute and the auxiliary operations to undertake.

The G commands or G codes are called preparatory functions and they prepare the machine controls for the type of machine movement which is going to take place. Just like G commands or G codes, there are M codes or commands which stand for Miscellaneous functions. Miscellaneous functions provide instructions to the machine about the auxiliary or non-cutting operations (like tool change, spindle start, coolant on etc.). They are sometimes referred to as managerial commands.

There are 4 basic types of motions for CNC machines. The rapid motion, the linear motion (at controlled feedrate), the circular clockwise motion & the circular counterclockwise motion.

The ordinary CNC machine can move fast from one point to another, cut straight lines and circles and practically nothing else. If it has to cut something more complex / intricate, the operator/ programmer has to break it up into simpler segments which can be approximated by circles and straight lines.

The motion commands are simple and short :

Code	Explanation	Format
G 00	Rapid traverse	G00 X 200 Y300 Z 400
G 01	Linear motion at controlled feed	G01 X 220 Y 320 Z440 F 200
G 02	Circular clockwise motion	G02 X 250 Y 350 R 150 F200 or G02 X 250 Y 350 CX 270 CY 370 F200
G 03	Circular counter clockwise motion	G03 X 300 Y 370 R 130 F 200 or

		G02 X 250 Y 350 CX 270 CY 370 F200
--	--	---------------------------------------

So, in the syntax of the G codes for motion, the target point has to be mentioned followed by other modifiers like F (feed in mm/min unless otherwise indicated), R (radius of circular interpolation) CX, CY (Center coordinates of the circle of interpolation) etc.

The initial point is generally the point at which the tool resides when the command is executed.

Explanation :

G00 makes the cutter move at the highest possible speed to the target point. The cutter might not actually execute the movements. In most cases, the table executes the X & Y movements while the spindle (holding the cutter, in case of vertical spindle machines) moves in the Z direction. G00 is generally used for locating/ positioning the cutter at some point, not for actual cutting.

G01 is used for taking linear cuts at rated feed value, mentioned in the command itself. For example, G01 X200 Y300 F250 would make the cutter move from the initial point (point of residence of the tool) to the point X200 Y300 with a velocity of 250 mm/min.

G02 and G03 are for taking circular cuts. G02 is for clockwise movements while G03 is for counter clockwise. G02 can occur along two paths from the initial point to the final point. So, yet another indicator is required for the uniqueness of the path.

Given the final and initial points, the radius of curvature and the sense of rotation (i.e., clockwise or counterclockwise), the cutter will move in a circular path satisfying all the applied conditions. If there are more than one path satisfying all the conditions, there would generally be a default path associated with the the command. For example,

If we consider the command :

G02 X20 Y30 R40 F100

And if the initial point is (15,25), there are two possible paths which satisfy all the conditions. In these cases, unless otherwise stated, the minor arc (the one which subtends a smaller angle at the center of the respective circle) movement would be chosen. If however, a negative sign precedes the magnitude of the circle radius, the major arc is selected.

G02 X20 Y30 R-40 F100

The Minor arc is chosen when no sign or a plus sign precedes the magnitude of the radius.

Now, the linear movement and the circular movements effected by G01, G02 and G03 respectively are obtained through a process called interpolation. Interpolation means that given the initial and final points of movement and the type of motion, the machine control unit essentially computes the intermediate positions and associated velocities of the cutter and imparts the respective axial velocities to the cutter so that it traces out the required path.