A study on basic preparatory and miscellaneous codes in FANUC control system for CNC machine for polytechnic students

k. Vinoth kumar[1]

[1]*Lecturer, Dept. of Mechanical Engineering, 219 Valivalam Desikar Polytechnic college, Tamil Nadu, India*

---***---

Abstract – In modern-day manufacturing process accuracy and time plays a vital role. Hence the manufacturing sectors moves towards deploying CNC. Mostly diploma students are employed as CNC programmers and operators. In this paper, basic preparatory and miscellaneous codes in FANUC control system are studied. This gives general idea about CNC FANUC control system for diploma students so that they can practice.

Key Words: CNC machine, FANUC control, preparatory code, miscellaneous codes.

1.Introduction

CNC stands for computer numerical control. In which the machine movements, operations are controlled by set of codes, that have syntax. Most of the CNC programs are coded in FANUC control system. This set of codes in total called as part program, this part program is a collection of words, characters, numerricals and are called as blocks. A set of blocks that describe a profile of a product is called as part program. Usually the part program contains codes, which are machine control and tool control.

1.1 Part program format

Part program is the set of instructions that are manually written by the programmer according to the job given in the blue print.

The part program format is given in the following table-1

Table-1

Symbol	Meaning
N	Block number
G	Preparatory code / function
X , Y, Z, A,B,C	Coordinates data / axes
F	Feed
S	Spindle speed
T	Tool number / designation
M	Miscellaneous codes/ functions

1.2 Basic terminologies

Sequence number- this is also called as block number and every program line starts with sequence or block number. This numbers are used while performing sub-program, mirroring and canned cycles.

Coordinate system- it is essential to apply coordinate system for machining in CNC machines. This is the process of identification and designating various axes in the machine. The axes may be vertical, horizontal and or traverse.

Datum points- these are the reference points for various operations, tool offset settings which is fixed by the manufacturer and by the programmer. Some of the datum points are machine zero, work zero, tool length reference plane and tool offsets.

CNC dimensioning- this deals with where program to be started and end with reference to work zero points. There are two types of dimensioning

- Absolute mode or fixed zero mode
- Incremental mode or floating zero mode

2. List of basic G codes, meaning and syntax

Some of the basic and important G codes, their meaning and syntax is given in the following table-2

Table-2

G code	Meaning	Syntax
G00	Rapid traverse- tool moves in maximum speed as set by the manufacturer- this command must be used when no machining is carried out.	**G00 X.....Y....Z....** X Y- end point of X Y coordinates
G01	Linear interpolation- tool moves in linear path	**G01 X....Y....** X Y – end point coordinates

G02	Circular interpolation-clockwise direction-the tool moves in a curved path in clockwise direction	**G02 X_1…Y_1..X_2…Y_2…R…F….** X_1 Y_1- end point of arc X Y coordinates X_2 Y_2 – centre point of arc XY coordinates R- radius of arc F- feed – to be reduced while performing arc
G03	Circular interpolation-counter clockwise direction- the tool moves in a curved path in counter clockwise direction	**G03X_1…Y_1..X_2…Y_2…R…F….** X_1 Y_1- end point of arc X Y coordinates X_2 Y_2 – centre point of arc XY coordinates R- radius of arc F- feed – to be reduced while performing arc
G20	All input values in INCH	Inch data
G21	All input values in MILLIMETERS	Metric data
G28	Return to home position	When this command is used the machine moves to home position as set by the manufacturer. Simply, This may be considered as reference point return command.
G40	Cutter [milling]/ tool nose [turning]compensation cancel	Cutter[milling] / tool nose [turning] compensation is cancelled
G41	Cutter[milling] / tool nose [turning] compensation LEFT	Cutter[milling] / tool nose [turning] compensation is provided on LEFT side.
G42	Cutter[milling] / tool nose [turning] compensation RIGHT	Cutter[milling] / tool nose [turning] compensation is provided on RIGHT side.
G98	Feed rate per minute	Mm/min
G99	Feed rate per revolution	Mm/revolution

2.1 List of basic M codes, meaning and syntax

Some of the basic and important M codes, their meaning and syntax is given in the following table-3

Table-3

M code	Meaning	Syntax
M 00	Program stop	Single command
M 01	Optional stop	Single command
M 02	Program end	Single command
M 03	Spindle ON clockwise	Single command
M04	Spindle ON counter-clockwise	Single command
M05	Spindle stop	Single command
M 06	Tool change	M06 T (tool number)
M 07	Coolant ON – low pressure	Single command
M 08	Coolant ON- high pressure	Single command
M 09	Coolant OFF	Single command
M 30	Program stop	Single command
M98	Sub program call	Single command
M 99	Sub program end	Single command

3. Important canned cycle commands for turning and milling module

Canned cycle:

Are those cycles that are specifically, employed for repetitive tasks such as drilling, profile turning.

3.1 List of canned cycles for turning and milling

Various canned cycles are listed in the following table.

Table-4: canned cycles

Code	Used in CNC Machine	Meaning / action	Syntax
G 70	Turning	Finishing cycle	**G70 .P..Q..F** P- start block Q- end block F- feed
G71	Turning	Multiple stock / rough removal cycle	**G71 U…R..** **G71 P..Q..U…W..** P- start block Q- end block

International Research Journal of Engineering and Technology (IRJET) e-ISSN: 2395-0056
Volume: 06 Issue: 05 | May 2019 www.irjet.net p-ISSN: 2395-0072

			U-finishing allowance in X W- finishing allowance in Z
G73	Turning	Pattern repeat cycle	
G74	Turning	Peck Drilling cycle	**G74R….** **G74Z..Q..R..F..** Z- total depth Q- depth/cut R- relief amount F- feed rate
G75	Turning	Grooving cycle	**G75 R….** **G75 X..W..Q..F..** R- retraction X- total depth W- width of groove Q- depth of cut F- feed rate
G78	Turning	Multiple turning cycle	**G78 P(m,r,a)…..Q…R…** **G78X…Z…P…Q..F..** m- repetitive count in finishing (1 to 99) r- chamfering amount a- Angle of thread Q- minimum cutting depth R- finishing allowance X – final depth of thread (minor diameter) Z- length of thread P-height of thread as radius value in microns Q- depth of thread in first cut as radius value in microns F- pitch
G80	Turning/ Milling	Canned cycle cancelled	Single command
G81	Milling	Drilling cycle	**G83 X..Y..Z..R..Q..F..** X,Y – coordinate points of drill Z- depth of hole R- reference point F- feed rate
G82	Milling	Counter sinking & counter boring cycle	

G83	Milling	Peck drilling cycle	**G83 X..Y..Z..R..Q..F..** X,Y – coordinate points of drill Z- depth of hole R- reference point F- feed rate
G84	Milling	Tapping cycle	
G85	Milling	Reaming cycle	
G86, G87, G88,G89	Milling	Boring cycle	
G170 G171	Milling	Circular pocketing cycle	**G170 R..P…Q…X..Y..Z..i…j..k…** **G171 P..S..R..F..B..j…Z…** R- tool position at start of cycle P- roughing Q- depth of cut/pass X,Y,Z- coordinate points of center of pocket i,j – finishing allowance R- radius of circular pocket P- cutter width percentage S- spindle speed during roughing R- feed in Z during roughing F- feed in X,Y during roughing B- spindle speed during finishing j- finishing feed.
G172 G173	Milling	Rectangular pocketing cycle	**G172 i..j…k…P…Q…R…X…Y…z…..** **G173j…k…P…T…S…R…F…B….j….Z….** i,j – dimension of pocket in X,Y direction k- corner radius P- roughing Q- depth of cut/pass R- absolute depth from surface. X,Y- pocket corner coordinates Z- base of pocket i,k-side and base finishing allowance

			P- cutter width percentage T- tool number S- spindle speed during roughing R- roughing spindle feed in Z F- roughing feed in X,Y B- spindle speed during finishing j- finishing feed

3.2 Special commands

Since CNC machines are employed for accurate, fast and complex shapes, there are special types of commands that can be used for reducing the machining, programming time and for error free operations. Some of such commands are listed below

- Mirroring
- Sub- programs / Sub- routines

sub programs: These are small programs that specifies the actual profile and machining path. They are stored separately and can be called in main program as and when required. They can be understood as mini-programs inside a main program.

Syntax

L1234 [sub program address]
………..
……….
………..[sub program statements]
…………
M99 [sub program end].

Calling sub-program in main program

Main program
………
……….
M98 L1234 P2 [calling sub program with address and number of repetitions as passes P]
Main program
……
…….

Mirroring

This command is employed in milling, in which, a profile in a single quadrant [for example, I quadrant] can be mirrored in II, III and IV quadrants with offset distances in X or Y or in both X&Y coordinates.

Syntax

M70 mirror in X ON
M71 mirror in Y ON
M80 mirror in X OFF
M81 mirror in Y OFF

While operating these commands, it has to be coordinated with respective sub program.

For example,
While performing mirror in X axix, the command sequence is
M70
M98 sub program address/ identity
M71
And the same sequence to be followed for mirror in Y axis ans so on.

4. CONCLUSIONS

In this paper, various basic commands used in a FANUC controlled CNC machine are studied with meaning and syntax in a lucid manner. This paper gives bird's eye view for beginners specifically for diploma students. By practicing these basic commands, they can develop basic knowledge about syntax and operations. More over the programing skills are enhanced for beginners pursuing diploma.

REFERENCES

[1] CAD/CAM/CIM by R.Radhakrishnan, S.Subramaniyan, New age international pvt limited.

[2] CAD/CAM by Miller.P.Groover , Emony Zimmers, Jr. prentice Hall of India pvt limited.

[3] CAD/CAM principles and applications, Dr.P.N.Rao, Tata McGraw Hill publishing company limited.

[4] Computer control of manufacturing systems, Yoram Korean, McGraw Hill book

[5] Automation, production systems and computer integrated manufacturing, Mikell P Groover, Pearson education Asia.

Author Photo