

Computer Numerical Control (Part-3)

8ME1A: Computer Integrated Manufacturing Systems

Tarun Kr. Aseri
Asst. Prof.
Mechanical Engineering
Engineering College, Ajmer
Barliya Chouraha, NH-8,
Ajmer-305025, India
Email: tarunaseri[at]ecajmer.ac.in



1

Outcomes

- Understand flow-process for CNC part program
- G and M Codes used frequently in CNC program
- Mode of CNC programming
- First manual CNC program using G & M codes



Tarun K. Aseri

Mechanical Engineering Department

2

1

CNC Words

- » **Block Number (N words):** N01, N05, N10
- » **Preparatory Code (G words):** G00, G01, G02, G03, G81, G98, etc.
- » **Coordinates (X, Y, Z words):** Movement in X, Y and Z directions
- » **Miscellaneous Code (M words):** M02, M03, M06, M08, M30, etc.
- » **Feed rate (F word):** inch/min or mm/min
- » **Speed (S word):** revolution per minute
- » **Tool number (T word):** Tool selection if ATC/magazine is available
- » **EOB: End of Block**

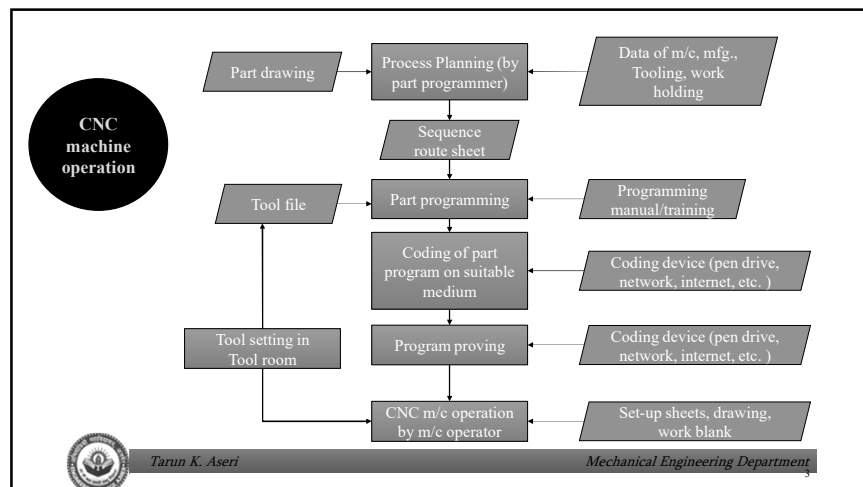


Tarun K. Aseri

Mechanical Engineering Department

4

3



Preparatory Codes

- » The term "preparatory" in CNC means that it "prepares" the control system to be ready for implementing the information that follows in the next block of instructions.
- » A preparatory function is designated in a program by the word address G followed by two digits.
- » Preparatory functions are also called G-codes and they specify the control mode of the operation.
- » G-codes assisted by dimensional parameters x, y, z, u, v, w, r and so on.



Tarun K. Aseri

Mechanical Engineering Department

5

Preparatory Codes: G Codes

G00	Rapid traverse	G50	Scaling
G01	Linear interpolation	G51/G52	Part rotation
G02/G03	Circular interpolation C.W/C.C.W.	G53-G59	Zero offsets
G04	Dwell	G63/G66	Programmable feed
G07	Tangential circle interpolation	G70/G71	Inch/metric dimensioning
G08/G09	Path control mode	G72/G73	Interpolation with in position stop
G10/G11	Block pre-processing control	G74	Home position
G12/G13	Circular interpolation with radius	G80-G89	Canned cycles
G17-G20	Plane selection	G90/G91	Absolute/incremental programming
G33	Thread cutting/rigid tapping	G92	Position register preset
G36/G37	Programmable feed Rate limitation	G94/G95	Feed Rate
G38/G39	Mirror image	G160-G164	ART learning function
G40-G44	Tool radius compensation	G186	Programmable tolerance band



Tarun K. Aseri

Mechanical Engineering Department

6

Miscellaneous Codes

- » Miscellaneous functions use the address letter M followed by two digits.
- » They perform a group of miscellaneous instructions/functions
- » For example coolant on/off, spindle on/off, tool change, program stop, or program end.
- » They are often referred to as machine functions or M-functions.



Tarun K. Aseri

Mechanical Engineering Department

7

Miscellaneous Codes: M Codes

M00	Program Stop.	M21	Mirror Image Along X Axis
M01	Optional Program Stop	M22	Mirror Image Along Y Axis
M02	Program End	M23	Mirror Image Cancel
M03	Spindle Motor On & Normal Clockwise	M30	Program End & Rewind.
M04	Spindle Motor On & Reverse Anticlockwise	M98	Subprogram Call with Program No. P
M05	Spindle Motor Off	M99	Subprogram End
M06	Automatic Tool Change		
M08	Coolant Pump Motor On.		
M09	Coolant Pump Motor Off		



Tarun K. Aseri

Mechanical Engineering Department

8

Programming Modes

Absolute programming (G90)

- In absolute programming, all measurements are made from the part origin established by the programmer and set up by the operator.
- Any programmed coordinate has the absolute value in respect to the absolute coordinate system zero point.
- The machine control uses the part origin as the reference point in order to position the tool during program execution

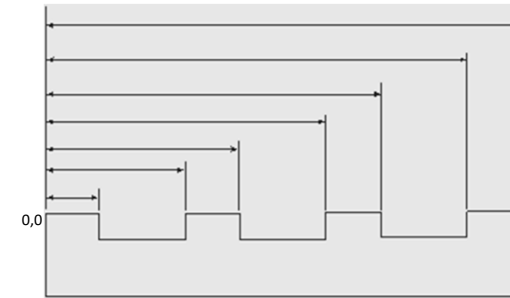


Tarun K. Aseri

Mechanical Engineering Department

9

Programming Modes: Absolute programming (G90)



Tarun K. Aseri

Mechanical Engineering Department

10

Programming Modes: Incremental programming (G91)

- In incremental programming, the tool movement is measured from the last tool position.
- The programmed movement is based on the change in position between two successive points.
- The coordinate value is always incremented according to the preceding tool location.
- The programmer enters the relative distance between current location and the next point.
- Also known as floating zero mode

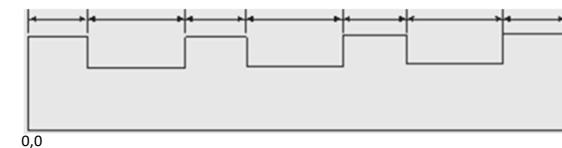


Tarun K. Aseri

Mechanical Engineering Department

11

Programming Modes: Incremental programming (G91)



Tarun K. Aseri

Mechanical Engineering Department

12

Rapid Traverse: G00

Syntax G00 Xx Yy Zz

Example

N10 G00 X10.0 Y15.0 Z2.0

Tarun K. Aseri
Mechanical Engineering Department

13

Linear Interpolation: G01

Syntax G01 Xx Yy Zz Ff

Example

N15 G01 X15.0 Y20.0

Tarun K. Aseri
Mechanical Engineering Department

14

Example

Tarun K. Aseri
Mechanical Engineering Department

15

Example

```

N1 G54
N2 G91
N3 T1 S1500 M03 F50
N4 G00 X20 Y20
N5 G01 Z-1 M08
N6 G01 X20 Y40
N7 G01 X20 Y-40
N8 G01 X-40
N9 G01 Z4
N10 G00 X-20 Y-20 M09
N11 M30
    
```

Tarun K. Aseri
Mechanical Engineering Department

16

Summary

- Eight CNC words likely to be used in program
- G90: for absolute mode of programing
- G91: for incremental mode of programing
- G00: for rapid traverse
- G01: for linear interpolation/traverse



Tarun K. Aseri

Mechanical Engineering Department

17



Thank you for your
kind attention

18