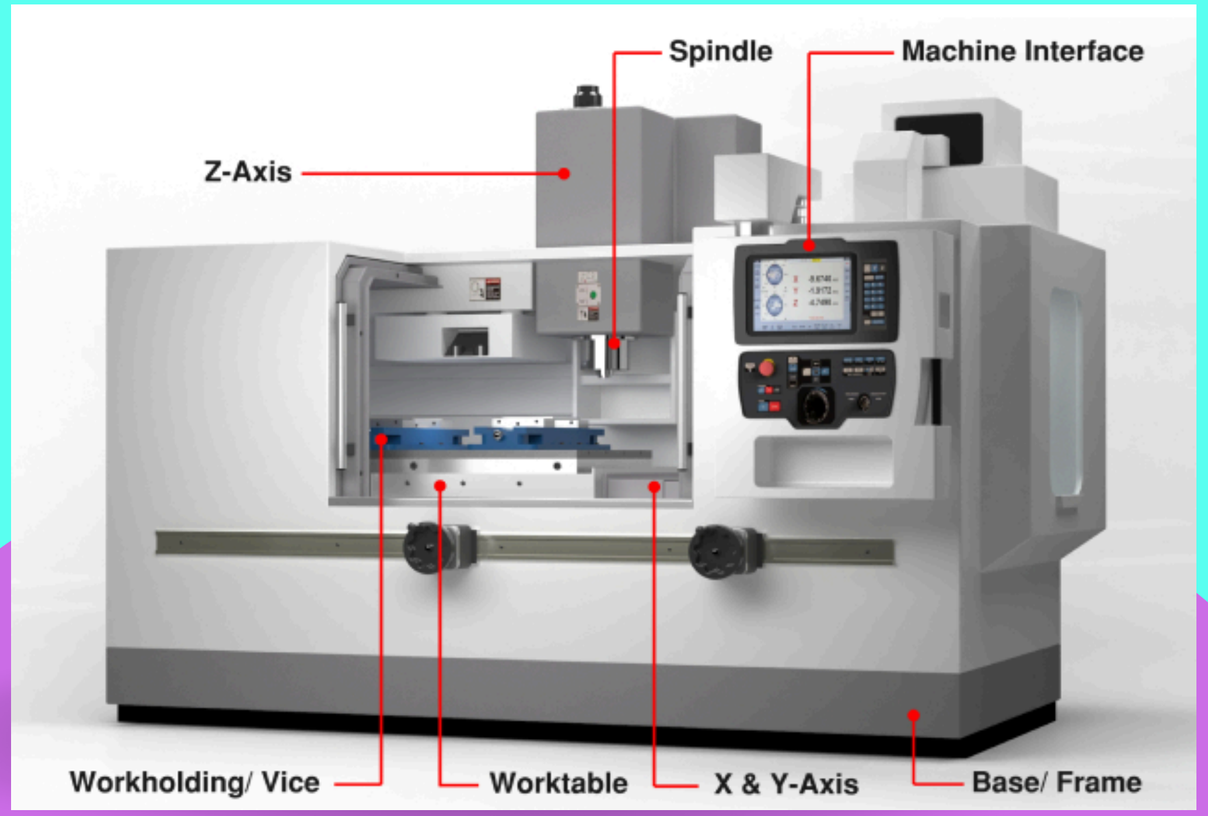
CODE-534

CLASS-1

unit -4

INTRODUCTION OF CNC MACHINE



Sparks Academy

81908 79379

G-CODE (Geometric Code)

These are used to control the movement and positioning of the machine tool.

Commonly Used G-Codes

G-Code	Function
G00	Rapid positioning (non-cutting movement)
G01	Linear interpolation (cutting in a straight line)
G02	Circular interpolation (clockwise)
G03	Circular interpolation (counter-clockwise)
G04	Dwell (pause for a specified time)
G17	Select XY plane
G18	Select ZX plane
G19	Select YZ plane
G20	Inch units

G21	Metric units
G28	Return to home position
G40	Cancel cutter radius compensation
G41	Cutter compensation left
G42	Cutter compensation right
G90	Absolute programming
G91	Incremental programming
G92	Set position
G94	Feedrate per minute
G95	Feedrate per revolution

M-CODE (Machine Code)

These control miscellaneous machine functions

M-Code	Function
M00	Program stop
M01	Optional stop
M02	End of program
M03	Spindle on (clockwise)
M04	Spindle on (counterclockwise)
M05	Spindle stop
M06	Tool change
M08	Coolant on
M09	Coolant off
M30	Program end and rewind
M98	Subprogram call
M99	Return from subprogram / loop

Canned Cycles (Cycle Codes)

Sparks Academy

These are predefined cycles for repetitive tasks like drilling, tapping, and boring.

G-Code	Function
G73	High-speed peck drilling cycle
G74	Left-hand tapping or boring cycle
G76	Fine boring cycle or threading cycle
G80	Cancel canned cycle
G81	Drilling cycle
G82	Drilling with dwell
G83	Peck drilling (deep hole drilling)
G84	Tapping cycle (right-hand)
G85	Boring cycle (feed in, feed out)
G86	Boring cycle (spindle stop, rapid out)
G87	Back boring cycle
G88	Boring cycle with dwell, manual retract
G89	Boring cycle with dwell, feed out

Ph. 81908 79379

End of Program Codes

Code	Function
M02	End of program (no rewind)
M30	End of program and rewind

Importance of Emergency Stop (E-Stop) Function in CNC Machine

The Emergency Stop (E-Stop) is a critical safety feature that immediately halts all machine operations to prevent injury, equipment damage, or tool breakage.

Key Points:

- **Instant Power Cut-Off: Disconnects power to drives/motors instantly.**
- **Used In: Emergency situations like tool collision, abnormal sound, program malfunction, etc.**
- **Location: Easily accessible red mushroom button on the CNC panel.**
- **Resets Safely: After resolving the issue, the machine needs to be reset manually to resume.**

Tool Offset in CNC Machine

Tool Offset is used to adjust the tool position to ensure accurate cutting without changing the actual program coordinates.

Types of Tool Offsets:

Type	Purpose
Length Offset (Z-axis)	Adjusts tool tip height based on tool length
Radius/Geometry Offset (X & Y)	Compensates for tool diameter/radius
Wear Offset	Adjusts for gradual tool wear during operations

Jog Mode and Its Role in Setting Tool Offset

Jog Mode allows manual movement of the machine axes to position the tool precisely.

How It Helps in Tool Offset:

- Select Jog Mode on the control panel.
- Use the manual axis control keys or dial to move the tool to the desired reference surface.
- Touch the tool to the workpiece surface (usually Z0).
- Set the tool length offset in the tool offset table for that tool number.
- Repeat for other tools using Jog Mode for accurate positioning.

Benefits of Jog Mode in Tool Offset:

- Precise manual control of each axis
- Prevents tool collision during setup
- Essential for setting reference positions
- Helps operators verify and set correct Z height

MDI Mode (Manual Data Input)

Definition:

MDI (Manual Data Input) mode allows the operator to manually enter and execute individual CNC commands, one line at a time, without writing a full program.

Key Features:

You can type G-codes, M-codes, or other instructions directly into the control.

Used for simple, quick tasks like:

- Spindle start/stop (M03, M05)
- Axis movement (G00 X50 Y50)
- Tool change (T01 M06)

Does not save the program; commands are temporary and executed instantly.

Example Use Cases:

- Manually moving the tool to a reference position
- Running a single drilling operation without loading a full program
- Turning the spindle on for manual inspection

Single Block Option (Single Block Mode)

Definition:

Single Block Mode allows the CNC machine to execute only one block (line) of the program at a time. After executing each block, the machine pauses, waiting for operator confirmation (usually via the Cycle Start button).

Key Features:

- Helps in troubleshooting, dry runs, and setup verification.
- Prevents unexpected tool crashes.
- Useful for debugging newly written or edited programs.
- Gives operator full control over program execution.

MDI and Single Block Work Together

- In MDI Mode, each command entered is treated like a single block.
- Therefore, Single Block mode is not typically used with MDI, because MDI already executes one block at a time.
- However, you can simulate similar control using MDI by:

1. Entering one command,
2. Observing the machine action,
3. Then entering the next command manually.

Feature	MDI Mode	Single Block Mode
Purpose	Manual entry of single commands	Execute program line-by-line
Usage	Setup, testing, manual operations	Debugging, dry run, testing
Control Level	High – enter only one command at a time	Very high – machine pauses after each block
Program Needed?	No	Yes (usually used with loaded program)

Edit Mode

- Allows the operator to open, create, modify, or delete CNC programs.
- You can change G-codes, M-codes, coordinates, or tool paths.
- Usually works with programs stored in memory or external sources (USB, card, etc.).

Key Functions:

- Add/delete lines in a program
- Rename or copy programs
- Correct tool offsets or feed rates
- Save updated programs

Auto Mode (Automatic Mode)

- Executes the full CNC program automatically.
- The machine follows the G-code instructions line-by-line without manual input.

Key Functions:

- Run stored programs from memory
- Used for production runs
- Executes motion, spindle, coolant, and tool commands automatically

Usage:

- After verifying and setting the program and tool offsets, run the job in Auto mode for full production machining.

Precautions

- Ensure safety: tool offsets, clamping, work zero, and dry run are verified before switching to Auto.

MPG Mode (Manual Pulse Generator / Handwheel Mode)

- Allows manual movement of machine axes using a handwheel or dial.
- Used for fine and precise positioning during setup.

Key Functions:

- Move X, Y, Z axes manually
- Set tool offsets or touch-off points
- Test small movements with high precision (e.g., 0.001 mm steps)

Feature	Edit Mode	Auto Mode	MPG Mode
Purpose	Edit or create program	Execute full program	Manual axis movement via handwheel
Execution	No actual machining	Full machining	Manual, fine movement
Used In	Program preparation	Production	Setup, tool alignment, touching off
Control Type	Manual programming	Automatic program execution	Manual movement control
Safety	Safe for edits (no motion)	High caution required	Safe but operator must watch carefully

Summary:

- Edit Mode → Program creation & modification
- Auto Mode → Full program execution
- MPG Mode → Precision manual movement for setup

1.what is the primary function of Edit Mode in CNC machines?

(A) To move the tool manually

(B) To execute a full program

(C) To modify or create CNC programs

(D) To reset tool offsets

2.Edit mode allows the operator to:

(A) Start spindle rotation

(B) Change or correct G-code lines

(C) Move axes automatically

(D) Switch tools during operation

3.What is the main purpose of Auto Mode in CNC operation?

(A) Create new program

(B) Check machine origin

(C) Manually move the tool

(D) Execute complete CNC program automatically

4.Auto mode executes commands stored in:

(A) Jog memory

(B) MDI

(C) Program memory

(D) Offset page

5.What does MPG stand for in CNC control?

(A) Machine Position Guide

(B) Manual Pulse Generator

(C) Machine Power Guide

(D) Main Program Generator

6.MPG mode is commonly used for:

(A) Running full programs

(B) Manual and fine tool positioning

(C) Editing feed rates

(D) Changing coolant direction

7.What does MDI stand for in CNC machines?

(A) Manual Data Input

(B) Machine Drive Interface

(C) Manual Directional Input

(D) Machine Direct Input

8.Which of the following is a suitable task in MDI mode?

(A) Continuous contour milling

(B) Full automatic machining

(C) Spindle start or stop

(D) Editing subprograms

9.Single Block mode allows the CNC machine to:

(A) Execute the entire program at once

(B) Run only one line/block at a time

(C) Perform threading cycle

(D) Change machine zero

10.Why is Single Block mode used?

(A) For full-speed machining

(B) To test and debug a program line by line

(C) To write macros

(D) To activate jog mode

11.What is a similarity between MDI mode and Single Block mode?

(A) Both are used for full program execution

(B) Both execute one block at a time

(C) Both save the program permanently

(D) Both are used to edit subprograms

12.Can Single Block mode be used with MDI?

(A) Yes, always

(B) No, MDI executes one block anyway

(C) Only in threading cycle

(D) Only in manual mode

13.What is the primary purpose of the Emergency Stop button on a CNC machine?

(A) To pause the program

(B) To save the program

(C) To immediately stop the machine for safety

(D) To reset the tool offset

1`4.Where is the emergency stop usually located?

(A) On the tool changer

(B) Near the spindle motor

(C) At an easily accessible place on the control panel

(D) Inside the electrical panel

15.Tool offset is used to:

(A) Increase spindle speed

(B) Adjust for tool wear or length

(C) Change tool number

(D) Stop coolant flow

16.Jog Mode in CNC machines allows the operator to:

(A) Write a program

(B) Edit tool offset values

(C) Manually move the axes

(D) Run automatic cycle

17.Which of the following is NOT done using Jog Mode?

(A) Manual tool touching

(B) Rapid automatic movement

(C) Setting reference zero

(D) Positioning tool manually

18.Tapping cycle is defined by which G-code?

(A) G81

(B) G82

(C) G84

(D) G86

19.Which cycle performs boring with feed in and feed out?

(A) G84

(B) G85

(C) G86

(D) G87

20.Which M-code stops the spindle?

(A) M30

(B) M05

(C) M01

(D) M02

Thank You