TNPSC >>> MECHANICAL

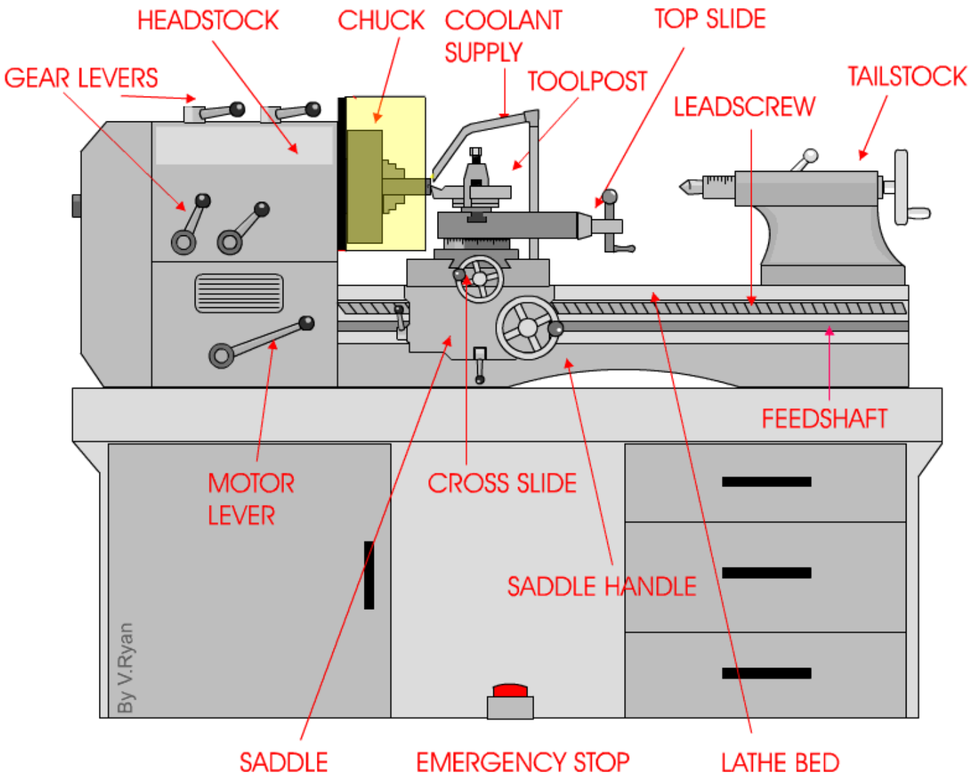
CNC & Numerical Control



Sparks Academy 81 908 79379

INTRODUCTION OF LATHE

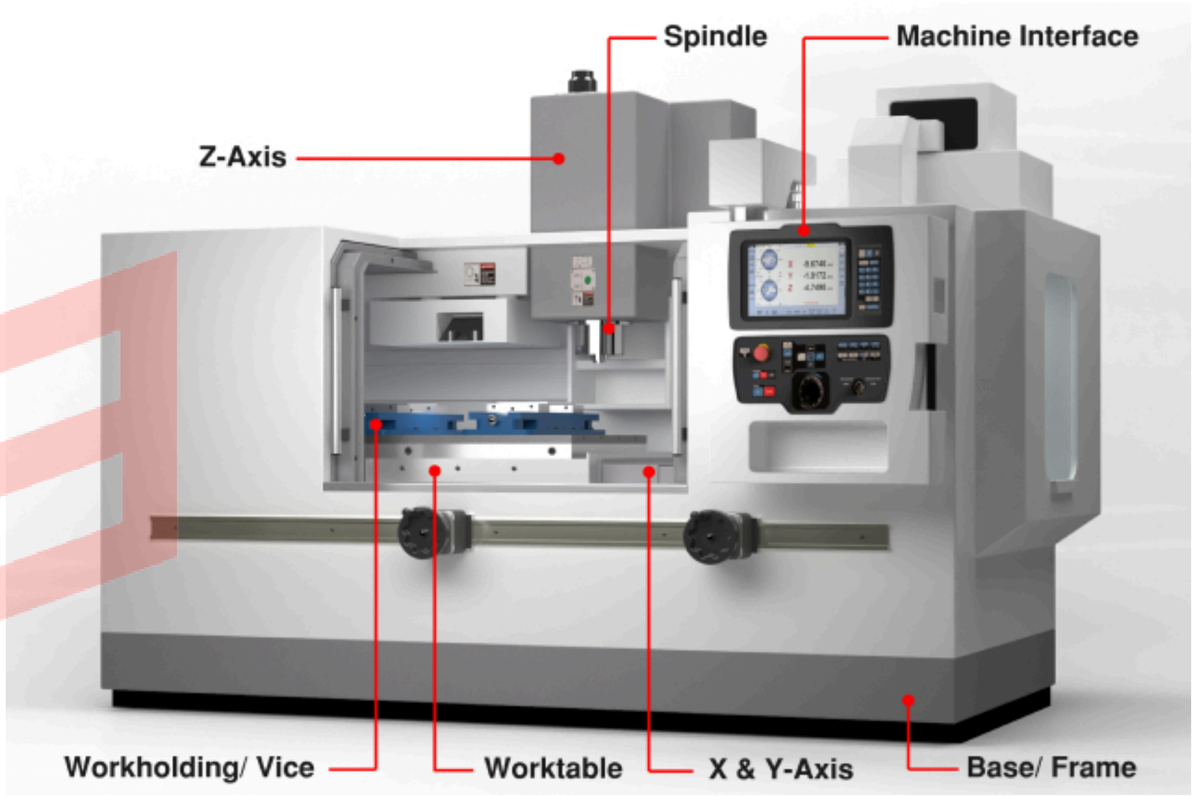
The primary task of a lathe is to generate cylindrical workpieces. The process of machining a workpiece to the required shape and size by moving the cutting tool either parallel or perpendicular to the axis of rotation of the workpiece is known as turning. In this process, excess unwanted metal is removed. The machine tool useful in performing plain turning, taper turning, thread cutting, chamfering and knurling by adopting the above method is known as lathe.



INTRODUCTION

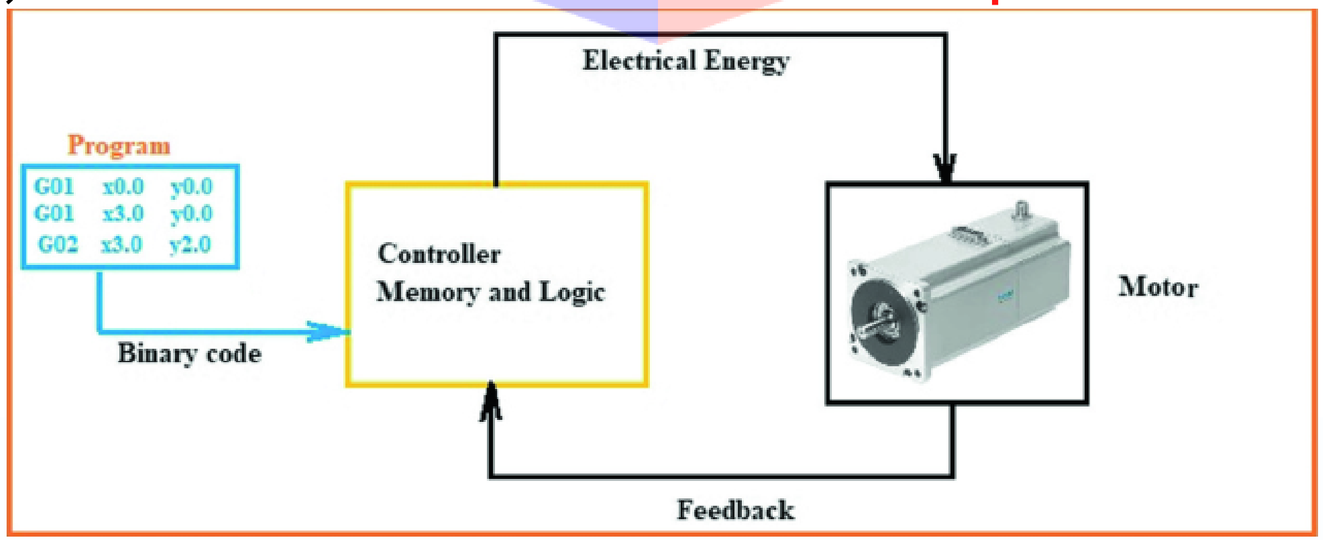
- CNC (Computer Numerical Control) Machines are automated machines, which uses programs to automatically execute a series of machining operations.

- CAD (Computer Aided Design) and CAM (Computer Aided Manufacturing) programs is used in modern CNC machines for end-to-end component design.

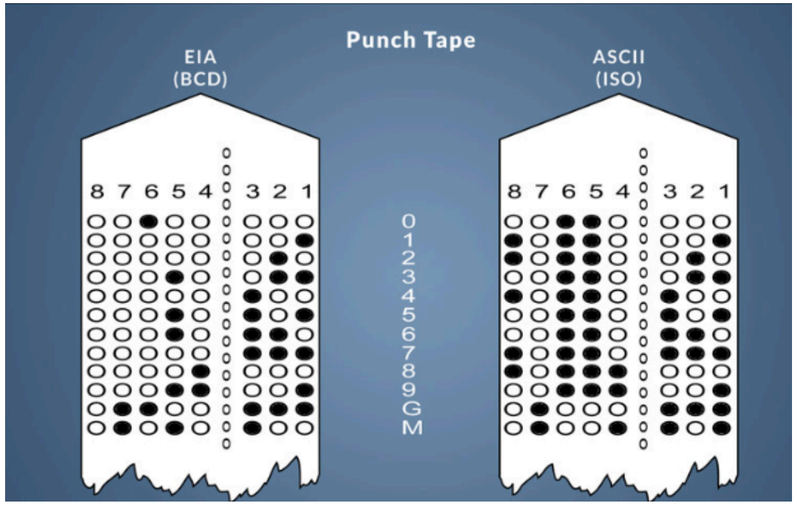


COMPUTER NUMERIC CONTROL MACHINE

- In CNC machine, program are designed or prepared first, then it is fed to the CNC machine.
- According to program, the CNC machine controls the motion and speed of the machine tools like (lathe, mills, shaper etc.).
- Feed drive in CNC milling machine are provided by servo motor.
- Feed motion in CNC drilling machine are provided by stepper motor.
- An interpolator provide two functions in CNC machine–
  - (i) It calculates individual axis velocity.
  - (ii) Drive the tool along the programmed path at the given feed rate.
- In a CNC machine tool, encoder is used to sense and control table position.

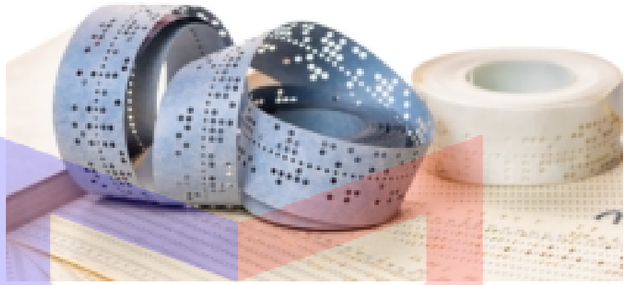


Main components of CNC machine



1. Input devices

- Input devices are used to input the part program in the CNC machine.
- Commonly used input device in CNC are :
  (a) Punch tape reader
  (b) Magnetic tape reader



2. Machine control unit (MCU)

- It performs all the controlling action.

3. Machine Tool

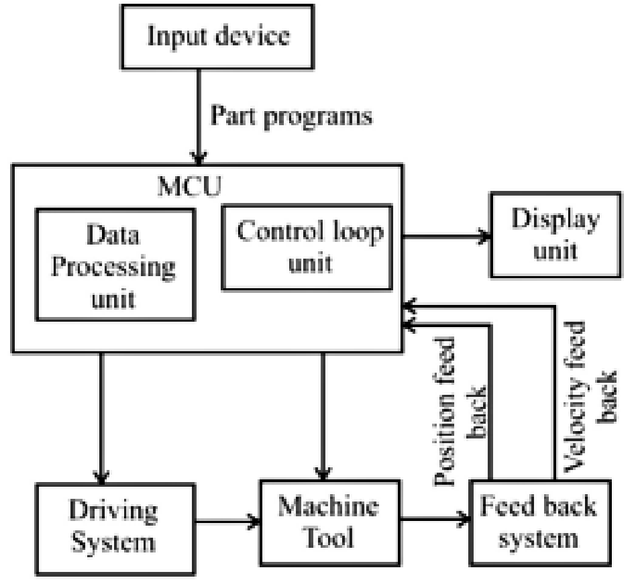
- A CNC machine tool always has a slide table & a spindle to control the position and speed

4. Driving system

- It consist of amplifier circuits drive, motor ball, lead screw.

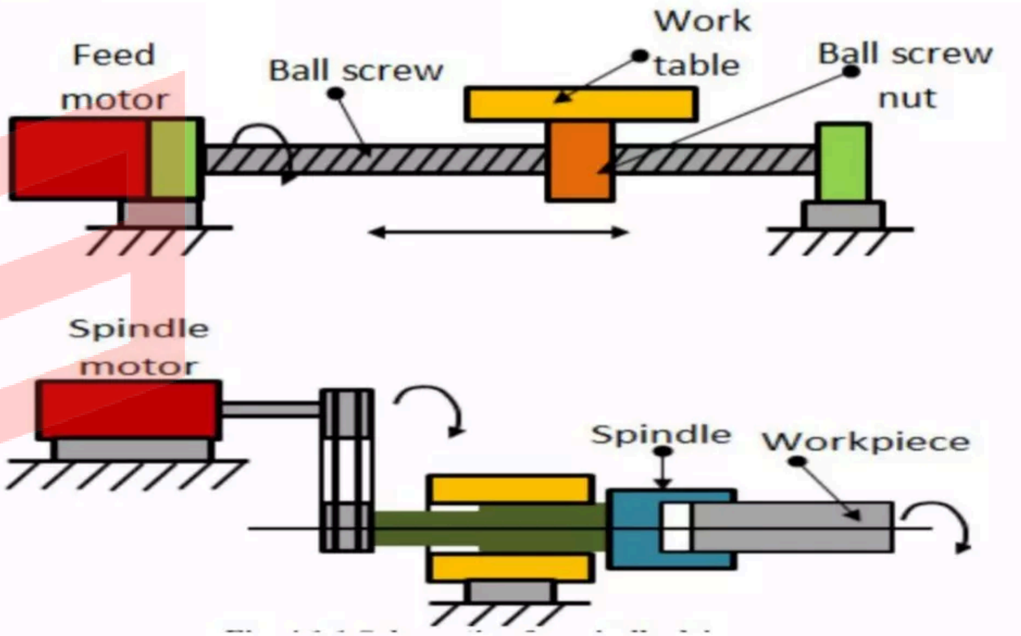
5. Feedback system

- It contains position & speed transducers that continuously monitor the position & speed of the cutting located at any instant.



DRIVING SYSTEM

- The driving system of a CNC machine consists of amplifier circuits, drive motors and ball lead screw.
- The MCU feeds the signals (i.e. of position and speed) of each axis to the amplifier circuits.
- The control signals are augmented (increased) to actuate the drive motors & the actuated drive motors rotate the ball lead screw to position the machine table.



Stepper Motor	Servo Motor
It operates in an Open loop System.	It operates in a Closed loop System.
Less Costly as Compare to Servo motor.	Servo motors are costly as compare to stepper motor.
It is small in size.	It is not as small.
It has no feedback system, so more prone to errors.	It has internal feedback system, less prone to errors.
It provides high amount of torque with low speed.	It provides less torque as compared to stepper motor.
Slow Speed as compared to servo motor.	It has higher speed than the stepper motor.
No need of encoder.	It requires encoder and gear box for more accurate control.
There will be no vibration or pulsation in stepper motors at standstill position.	Servo motors tends to pulsate or vibrate in standstill position.

ATC - AUTOMATIC TOOL CHANGER

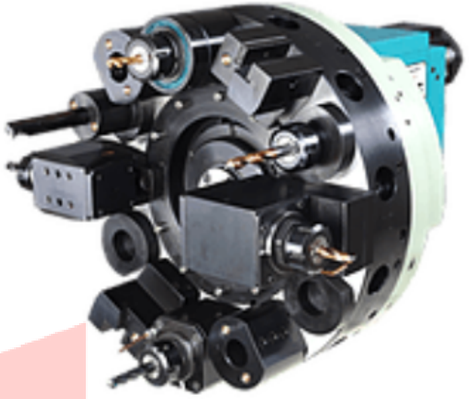
- ATC stands for automatic tool changer, which is a feature in computer numerical control (CNC) machines that can quickly change tools without human intervention.
- ATCs can improve a CNC machine's production and tool-carrying capacity, and they can be used to replace worn or broken tools.
- The time required to change a tool in a CNC Machine using ATC is around 3- 7 Seconds.



Here are some benefits of an ATC:

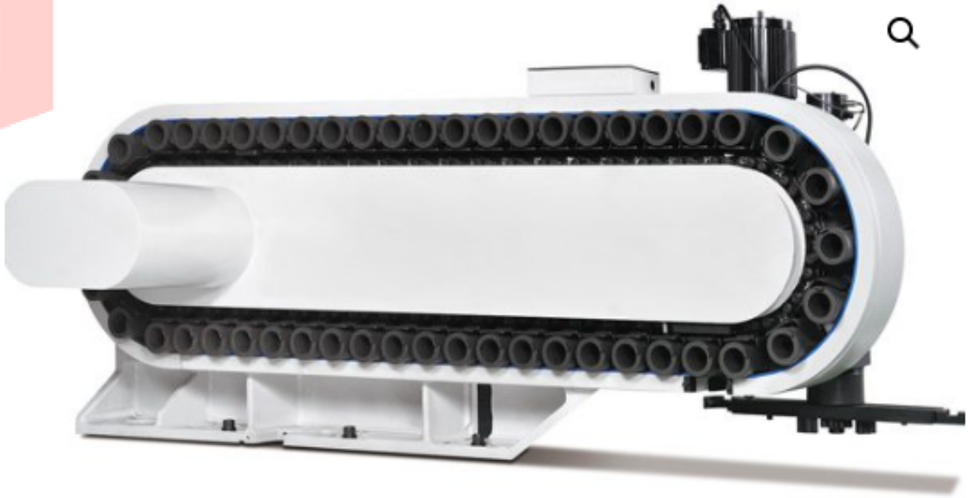
- Speed and accuracy: ATCs can increase speed and accuracy.
- Reliability: ATCs can increase reliability.
- Idle time: ATCs can reduce idle time.
- Ease of use: ATCs can make it easier to change large or heavy tools.
- ATCs are made up of several parts, including: A base, Arms with a gripper, A tool holder, A support arm, and Tool magazines.

TYPES OF ATC





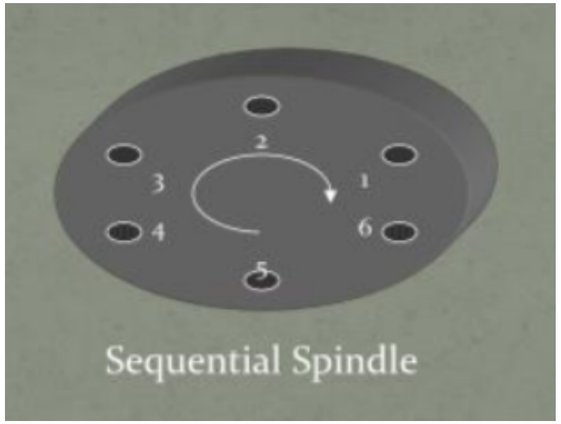
1. Turret head type ATC
   - It is generally used in CNC drill machine.
2. Drum type ATC
   - In this 30 tools are stored along the circumference of drum. Tool finding speed of this type of ATC is fast.
3. Chain type ATC
   - In this ATC more than 30-40 tools are store in it, having slow speed of tool finding. Double gripping system of tool changer.



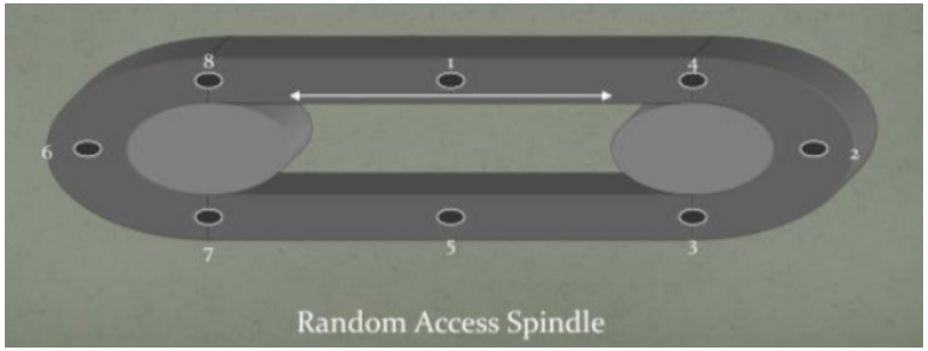
Tool Changer

Types

1. **Automatic (Sequential) Spindle**
   - Turns one increment in one direction for each tool change; tools must be placed in the spindle in the order they are used
2. **Indexable (Random Access) Spindle**
   - Turns either direction to make a specific tool accessible; tools can be placed in any order as long as the computer knows their positions



Sequential Spindle



Random Access Spindle

Automatic Tool Changer Mechanism

- After receiving the tool change command, the tool to be changed will assume a fixed position known as the "tool change position".
- The ATC arm comes to this position and picks up the tool.
- The arm swivels between machine turret and magazine.
- It will have one gripper on each of the two sides.
- Each gripper can rotate 90°, to deliver tools to the front face of the turret.
- One will pick up the old tool from turret and the other will pick up the new tool from the magazine.
- It then rotates to 180° and places the tools into their due position.

Advantages of Automatic Tool Changer

- Increase in operator safety by changing tools automatically
- Changes the tools in seconds for maintenance and repair
- Increases flexibility
- Heavy and large multi-tools can easily be handled
- Decreases total production time.

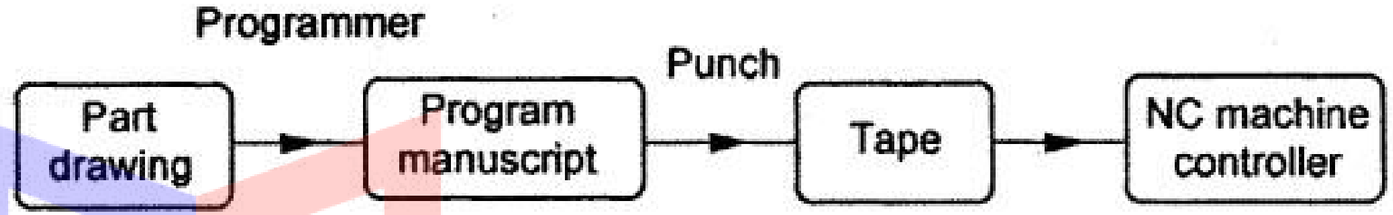
PART PROGRAMMING

Part program: It is a set of step by step instruction to the machine tool for carrying out the operation.

There are two method for part programming

(i) Manual part programming

(ii) Computer aided part programming



MANUAL PART PROGRAMMING:

- In this method, the programmer writes the Machining instructions on a special form called part program sheet.
- The program is directly input into the memory of the machine from a computer system in to the CNC memory through the keyboard of the CNC console.

Part program structure format

- Part program contains set of block. Each block contains set of NC words.
- Each NC word is the collection of characters used to form an instruction.
- (X position. Y position, speed etc are NC words).

PART PROGRAMMING FORMAT

Character : Meaning

A - Rotation about X-axis
B - Rotation about Y-axis.
C - Rotation about Z-axis
D&E - Rotation about additional axes
F - Feed rate
G - Preparatory function, identifying the action to be executed
H - Unassigned
I - Interpolation parameter/Thread pitch parallel to X-axis
J - Thread pitch parallel to Y axis
K - Thread pitch parallel to Z axis
L - Unassigned
M - Auxiliary function

Character : Meaning

N - Block number
O - Not used
P,Q,R - Thread movement parallel to X, Y & Z axes respectively. P & Q are also used as parameters in cycles.
S - Spindle speed
T - Tool
U,V,W - Second movement parallel to X,Y,Z axes respectively
X - Movement in X-axis
Y - Movement in Y-axis
Z - Movement in Z-axis

COORDINATE SYSTEM

- Machining of a workpiece by CNC requires a a Coordinate system to be applied to the machine tool.
- As all machine tools have more than one slide , it is important that each slide is identified individually.
- There are three planes in which movement can take place.
    - Longitudinal
    - Vertical
    - Transverse
- Each plane is assigned a letter and is referred to as an axis i.e. axis X axis Y and axis Z. The three axes are identified by uppercase X. Y. and Z and the direction of movement along each axis is specified as either plus or minus.
- Basically CNC lathe is specified by only two axes X and Z.
- For CNC milling and drilling machines, two axes X and Y are defined in the plane of table.
- Z axis is the vertical motion of the spindle.
- The Z axis is always the main spindle axis of the machine

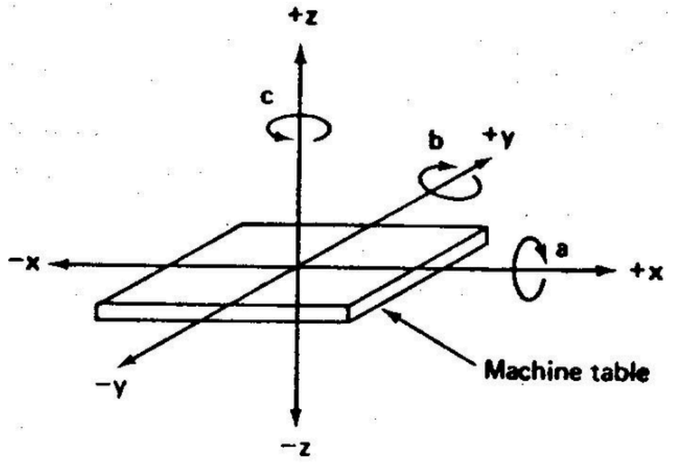


FIGURE 8.2 Machine tool coordinate system for NC.

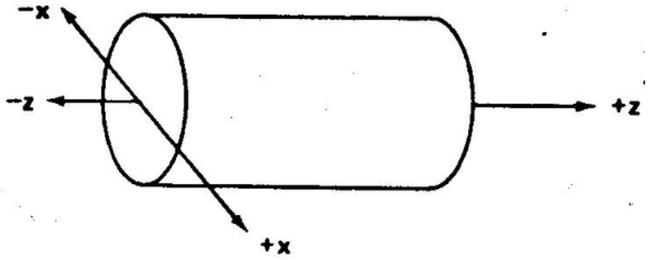
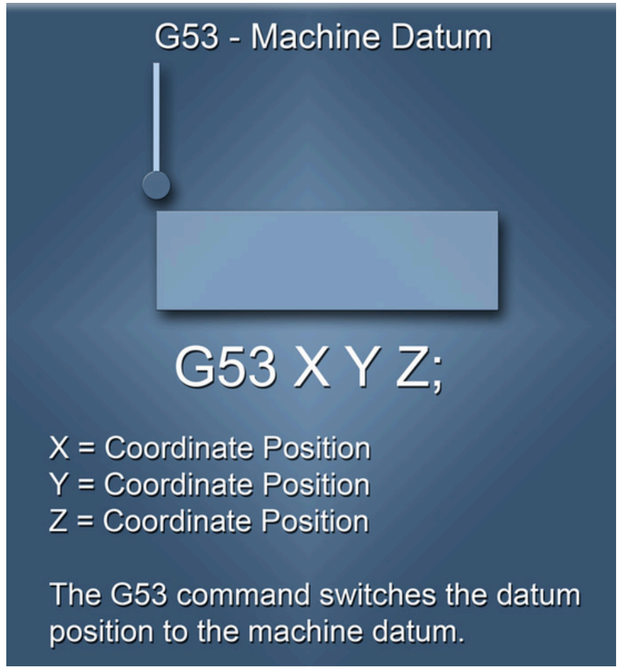


FIGURE 8.3 x- and z-axes for NC turning.

DATUM POINTS

- The datum point is the reference points assigned by the machine manufacturer for machine and programmer for the tool which help for exact positioning of tool on the work piece and also to control the movement of tool while machining a component.



ZERO POINTS

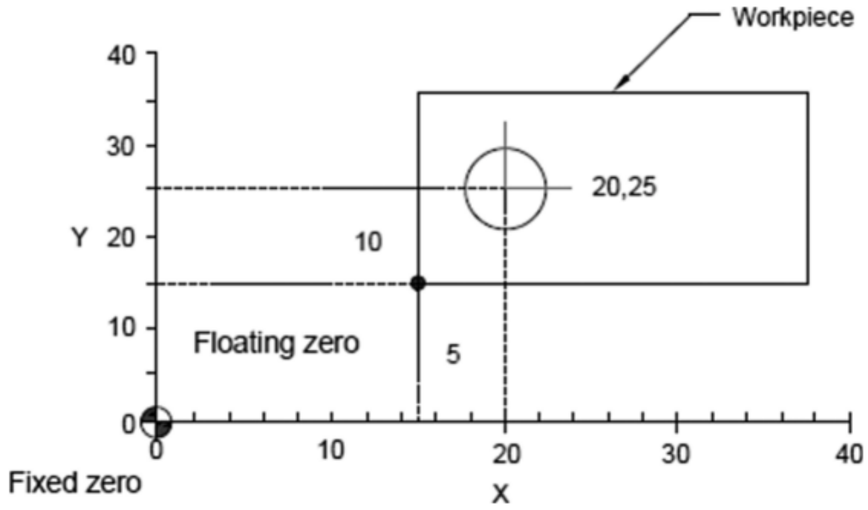
- It is the origin point of the co-ordinate system of the CNC machine tool.
- With respect to this origin point, programmer decides the tool position and movements.

There are two methods of specifying this zero point.

- Fixed zero
- Floating zero

Fixed Zero

- It is the zero point of the CNC machine tool fixed by the manufacturer.
- It is always located at the same position of the machine tool.
- For drilling, boring, milling and similar machines it is at the lower left corner of the machine table and
- For turning machines it is on the middle of the spindle flange.
- It is useful for programming with respect to the machine tool.

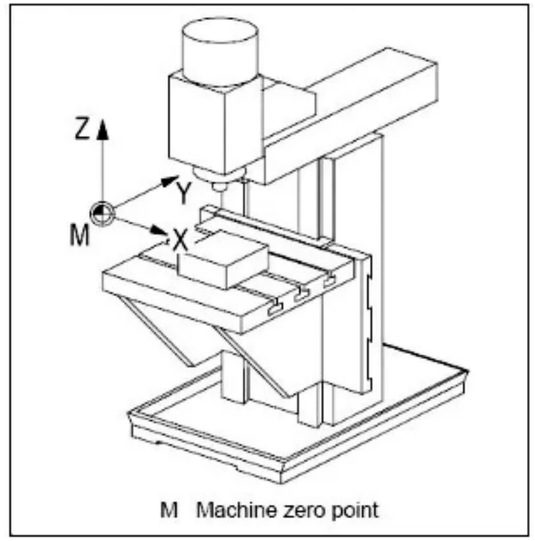


Floating zero

- It is the zero point of the NC machine tool set by operator at any position on the machine table/work piece.
- The part programmer decides this zero point depending on his programming suitability and communicate the same to the operator. Operator set this floating zero before starting the operation.

Machine Zero

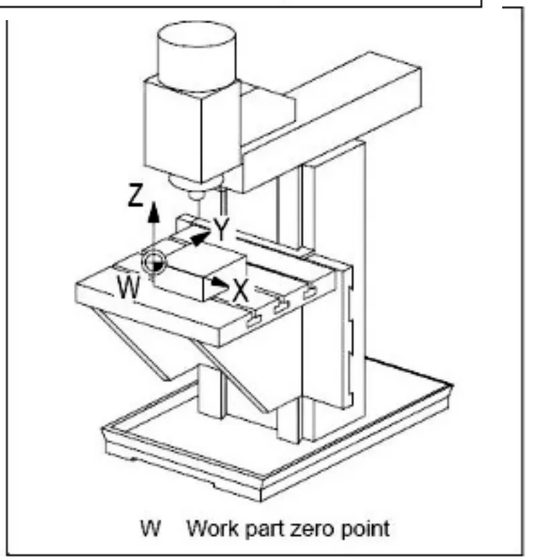
- Machine zero is a fixed point, set by machine manufacturer, it cannot be changed,
- The controller always remembers tool distance from the machine zero,
- Stored in offset register.



M Machine zero point

Work Zero

- In CNC, work zero is the origin of a part or stock's coordinate system, and it's also known as program zero.
- It's the position that's used as the starting point for a specific part program.
- This point is chosen by the part programmer and input to be machine controller.
- It is known as program zero point.



W Work part zero point

TYPES OF MOTION CONTROL

The three basic types of motion control in CNC are rapid, straight line, and circular:

Rapid motion:

- Also known as positioning motion, this is the fastest speed at which the machine can run. It's often used to move the tool to and from cutting positions, or to clear obstructions.

Straight line motion:

- This motion moves the tool in a straight line, and is often used for cutting, drilling, or milling straight surfaces.

Circular motion:

- This motion generates a consistent radius during machining.

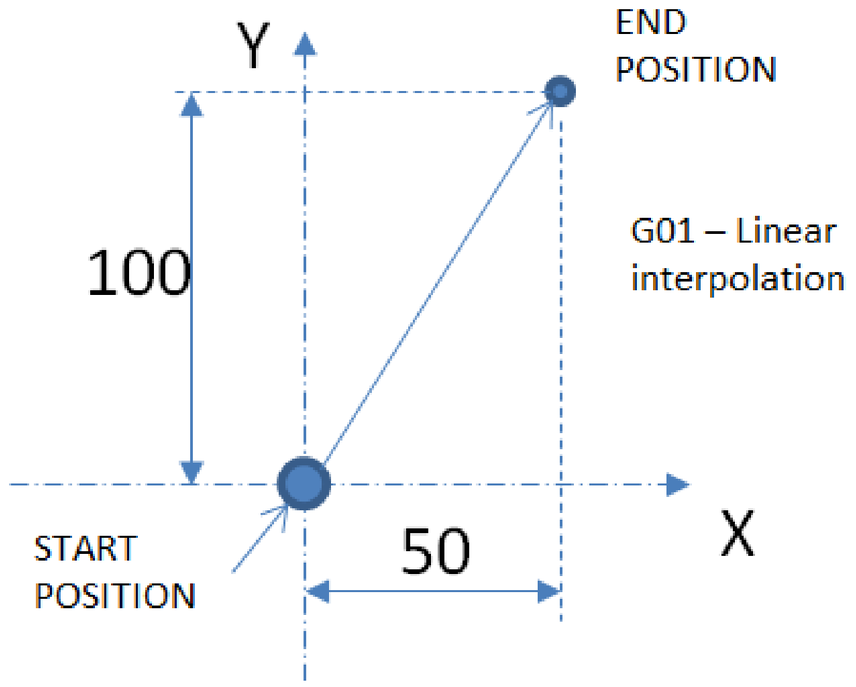
The amount of motion and the rate at which the axes move are all computer-controlled.

Types of Interpolation

- Linear interpolation.
- Circular interpolation.

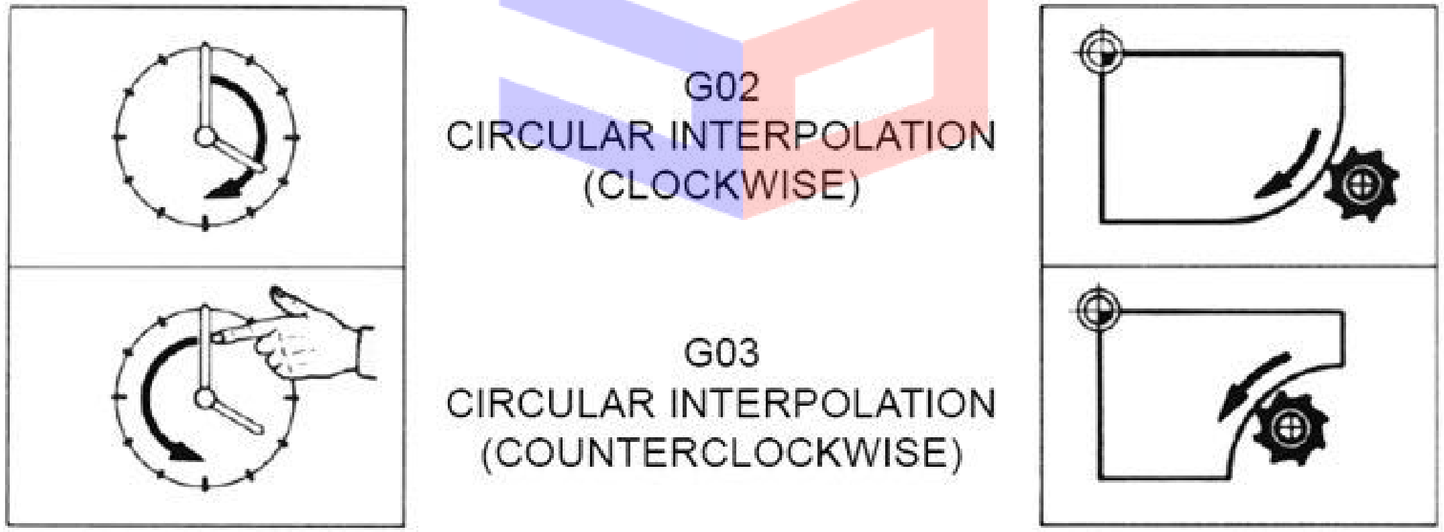
Linear Interpolation: G01

- Format: N_ G01 X_Y_Z_F_.
- Linear interpolation is a CNC machine movement that moves a tool from one point to another along a straight line.
- It's also known as straight cutting, and is one of the most common movements in CNC part processing.



Circular Interpolation: G02,G03

- Circular interpolation in CNC machining is the process of generating a circular motion by coordinating the movements of two axes.
- It uses the G02 and G03 commands, which work on both lathes and machining centers
- In CNC machining centers with three or more axes, you need to select the axis for circular interpolation using the G17, G18, and G19 codes.



G-CODE

- G-code, or geometric code, is a programming language used to control computer numerical control (CNC) machines.
- G-code commands are typically a combination of letters and numbers, and each command has a specific function.
- G-code is used in a variety of applications, including CNC mills, lathes, machining centers, jig borers, drills, and 3D printing.

Important G Code

Code	Function of code
G 00	Rapid movement
G 01	Linear Interpolation
G 02	Circular Interpolation (CW)
G 03	Circular interpolation (ACW)
G 04	Dwell
G 05	Spindle stop
G 17	XY plane designation
G 18	ZX plane designation
G 19	YZ plane designation
G 20	Clockwise circular interpolation for long dimensions
G 21	Input values specified in millimeters
G 28	Automatic return to reference point
G 32	Code for constant lead threading on CNC lath machine

Code	Function of code
G 33	Thread cutting
G 40	Cutter compensation cancel
G 41	Cutter radius compensation left offset
G 42	Cutter radius compensation right offset
G 43	Tool length compensation (plus)
G 44	Tool length compensation (minus)
G 49	Tool length compensation cancel
G 80	Cancel canned cycles
G 81	Drilling cycle
G 90	Absolute positioning
G 91	Incremental positioning
G 92	Absolute present, change the datum position

M CODE

- M-code, or miscellaneous function or auxiliary function code, is a type of code used in CNC (computer numerical control) programming to control non-movement related functions on the machine.
- M-code is used in conjunction with G-codes, which control the machine's movement, to create a complete CNC program.
- It controls miscellaneous machine functions, including starting and stopping specific actions or programs.

Important M Code

Code	Function of code
M 00	Program stop
M 01	Optional stop
M 02	End of program
M 03	Spindle CW
M 04	Spindle CCW
M 05	Spindle stop
M 06	Tool change
M 07	Tap oil on
M 08	Coolant on
M 10	Clamp on
M 11	Clamp off

SUBPROGRAM

- Subprograms are mini programs used in manual part programming.
- If the same machining operation, which was carried out already is to be performed at many different positions on the workpiece, it can be executed by Means of a program called as subroutines.
- Whenever a particular feature is required within the programme, the associated subroutine is called for execution.
- The subroutines may be called any time and repeated for 999 times.
- After execution of subroutine the control returns to main programme.
- The M98 command is used to call a subprogram, followed by the program number and the number of times to repeat it.
- Subprograms are terminated with M99, which returns the program to the main file.

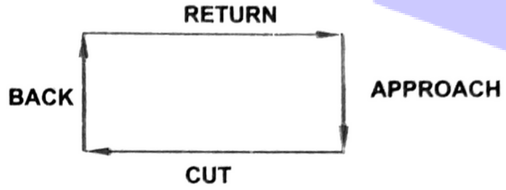
Ex. M98 P003444

- 003-The sub program has to be executed for three times.
- 444-name of the subprogram stored.
- M99- is the code for subprogram end and return to main program

CANNED CYCLES or FIXED CYCLES:

- The routine that automatically generates multiple tool movements from a single block is known as canned (or) fixed cycle.
- Canned cycle may be defined as a set of instructions, inbuilt or stored in the system memory, to perform a fixed sequence of tool movements.
- The canned cycle be brought into action with a single command.
- Canned cycles are used for repetitive and commonly used machining operations.

Eg): In turning, to take one depth of cut the tool makes the following movements.



- For the above movements program requires 4 blocks of commands.
- But by using canned cycle routine, the same is possible by single block of command.

List of Canned cycle codes

G code	Function
G71	Multiple/stock removal turning cycle
G73	Pattern repeating cycle
G74	Drilling cycle
G75	Grooving cycle
G76	Multiple thread cutting cycle
G80	Canned cycle cancel
G81	Drilling cycle
G82	Counter sinking and counter boring cycle
G83	Deep hole drilling cycle
G84	Tapping cycle
G85	Reaming cycle
G86,87,88,89	Boring cycle
G170, G171	Circular pocketing cycle
G172, G173	Rectangular pocketing cycle

➤ **Feed drives in CNC milling machines are provided by**

(A) synchronous motors **(B) induction motors**

(C) stepper motors **(D) servo-motors**

➤ The part programming code G90 signifies _____.

(a) Input values specified in inches.

(b) Programming in incremental coordinates.

(c) **Programming in absolute coordinates.**

(d) Input values specified in millimeters.

(e) Answer not known

➤ Which code is used to cancel the canned cycle

(a) G81

(b) G82

(c) G85

(d) **G80**

(e) Answer not known

➤ Which of the following is not a basic component of the Numerical Control System?

(a) machine control unit
(b) part program
(c) **part programmer**
(d) processing equipment
(e) Answer not known

➤ The CNC part programming code G32 represents

(a) Exact Stop
(b) **Thread cutting**
(c) Tool-nose radius compensation left
(d) Tool-nose radius compensation cancel
(e) Answer not known

➤ The function of a Tachogenerator in CNC machine is

(a) **Faster Response of Motors**
(b) Reduction of Basic Length unit
(c) To make 3-D interpolation possible
(d) None of the above
(e) Answer not known

➤ Numeric control

(a) is a method for producing exact number of parts per hour
(b) **is a method for controlling by means of a set of instructions**
(c) applies only to milling machines
(d) All the above
(e) Answer not known

38. M codes in CNC par programs are

(A) Switching functions

(B) Measurement functions

(C) Preparatory functions

(D) Compensation functions

37. In G82-Drilling or counter boaring cycle, the letter 'K' refers to

(A) Distance from I to R level

(B) Dwell time

(C) Feed rate

(D) No. of repeats

135. In the CNC part program, the function G01 represents

(A) Rapid Positioning

(B) Linear Interpolation

(C) Circular Interpolation CW

(D) Circular Interpolation CCW

196. In CNC Electric Discharge Machining, the gap between the electrode and work piece is in the range of

(A) 0.006 mm to 0.06 mm

(B) 0.004 mm to 0.04 mm

(C) 0.005 mm to 0.05 mm

(D) 0.007 mm to 0.07 mm

197. In CNC Electric Discharge Machining, the amount of heat produced due to spark is about

(A) 1100°C

(B) 1200°C

(C) 1500°C

(D) 1700°C

198. The time required to change a tool in a CNC Machine using ATC is around

(A) 4 – 6 Seconds
(C) 1 – 3 Seconds
(C) 3 – 7 Seconds
(D) 8 – 10 Seconds

199. In CNC Machine, the principle motion of cutting tool or the work piece designated by

(A) A Axis
(C) Y Axis
(B) X Axis
(D) B Axis

200. In CNC Programming G76 is for

(A) Finish turning canned cycle
(B) Multiple facing cycle
(C) Multiple thread cutting cycle
(D) Peck drilling cycle

195. In CNC system, MCU stands for

(A) Modem Computer Unit
(B) Machine Computer Unit
(C) Machine Control Unit
(D) Modem Control Unit

37. The feedback device used to measure the absolute position reached by the slide, in a direct way is called as

(A) Absolute encoder
(B) Incremental encoder
(C) Linear scale
(D) Tacho generator
(E) Answer not known

39. The preparatory function used to execute the cutter radius compensation offset left is given by

(A) G40
(B) G41
(C) G42
(D) G80
(E) Answer not known

40. The feed drive motors usually used in open-loop control systems are

(A) AC Servomotor
(B) DC Servomotor
(C) Stepper motor
(D) Linear motor
(E) Answer not known

136. In CNC machines, the secondary slide motions are labeled as ———— as per EIA standards.

(A) X, Y and Z
(B) U, V and W
(C) a, b and c
(D) A, B and C
(E) Answer not known

137. The type of tool magazine used to store a large number of tools, having high flexibility in operation and can be duplicated is known as

(A) Turret type magazine
(B) Drum type magazine
(C) Disc type magazine
(D) Chain type magazine
(E) Answer not known

138. IN CNC turning centers, the diameter programming is mostly used to define the movement along X axis, the tool approaches for machining only

(A) Radially
(B) Diametrically
(C) Diagonally
(D) Spherically
(E) Answer not known

140. The part program word, N075 G00X0Y0 Z50 ; means

(A) Circular interpolation clockwise

(B) Circular interpolation anticlockwise

(C) Linear interpolation

(D) Rapid movement to set point

(E) Answer not known

141. In CNC machines, the faster movement of slides are achieved by

(A) V-shaped slide ways

(B) Linear scales

(C) Rotary encoding systems

(D) Linear motion systems

(E) Answer not known

THANK YOU